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AUTOCAD® 2010 ESSENTIALS

COMPREHENSIVE EDITION

Munir M. Hamad

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COMPREHENSIVE EDITION

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Autodesk® Approved Instructor



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PREFACE

INTRODUCTION

- ◇ Autodesk® AutoCAD® has been the de facto drafting tool for PC users since 1982. As you read this, millions of engineers, draftsmen, project managers, and engineering students are creating their drawings with AutoCAD.
- ◇ This book introduces new and novice users to AutoCAD 2010, covering both Level 1 and Level 2 concepts. It is also a very handy tool for college and university drafting instructors using AutoCAD 2010.
 - Currently, about 75% of current AutoCAD 2010 users regularly apply the core Level 1 commands for AutoCAD 2010.
- ◇ This book will not teach what engineering drafting is or how to produce it; knowing drafting and design concepts are prerequisites for using this book.
- ◇ This text can be instructor-led or self-taught.
- ◇ At the end of each chapter, you will find Chapter Review questions that will help you practice and test your mastery of the material.
- ◇ There are 94 exercises integrated throughout the book to help you quickly implement what you have learned. The first half of the book contains 21 workshops that, together, will complete a full project (a small villa), starting with the creation of the project through plotting. Solving all 94 exercises will teach you to:
 - Simulate a real-life project from beginning to end, thereby allowing you to implement what you have learned.
 - Learn all of the basic commands and functions in AutoCAD 2010.
 - Apply advanced Level 2 concepts in real-world scenarios.

PURPOSE AND OBJECTIVES

- ◇ At the completion of this book, readers will be able to:
 - Understand AutoCAD 2010 and its basic operations, including the filing system
 - Draw different objects with speed and precision
 - Set up drawings and construct them in simple steps
 - Modify any object in a drawing
 - Create, insert, and edit blocks
 - Hatch using different hatch patterns and methods
 - Create text and tables
 - Insert and edit dimensions
 - Prepare and plot a drawing
 - Create dynamic blocks
 - Create, edit, and extract block attributes
 - Use the External Reference feature to collaborate with teams
 - Understand sheet sets
 - Use basic, intermediate, and some advanced layer functions
 - Utilize Autodesk Design Review software for markups
 - Use the AutoCAD Markup Set Manager

PREREQUISITES

- ◇ The author assumes that you have experience using computers and the Microsoft® Windows® operating system.
- ◇ Also, readers should know how to start new files, open existing files, save files, use “Save As” with files, close files with or without saving, and exit software.
- ◇ Because these commands are similar in all software packages, the author does not cover these subjects unless it is necessary to demonstrate a command specific to AutoCAD.

ABOUT THE DVD

- ◇ A DVD is included with the book and contains the following:
 - The AutoCAD 2010 trial version, which will last for 30 days starting from the day of installation. This version will help you solve all of the exercises and workshops in the book. Students with a valid university email address can visit <http://students8.autodesk.com/?lbon=1> for student versions of the AutoCAD material for the duration of the class. After successfully downloading the trial version from the DVD and starting AutoCAD 2010, users will be prompted by an activation menu. Select “Run the product” to begin working with this trial software. To activate your trial version, use product key 001B1 when prompted by AutoCAD.
 - Exercise and workshop files, which will be your starting point to solving all exercises and workshops in the book. Copy the Book Exercises and Book Workshops folders onto your hard drive. You will find two folders inside the Book Workshops folder. The first one is named “Metric” for the metric units workshops, and the second one is named “Imperial” for the imperial units workshops.
 - Selected solutions to exercises and workshops.
 - All images from the text in full color.

INTRODUCTION TO AutoCAD® 2010

In This Chapter

- ◇ What Is AutoCAD®?
- ◇ How to Start AutoCAD® 2010
- ◇ Understanding the AutoCAD® 2010 Interface
- ◇ Points in AutoCAD®
- ◇ AutoCAD® Default Settings
- ◇ Things You Should Know about AutoCAD®
- ◇ Drawing Limits
- ◇ Units and Spaces
- ◇ Viewing Commands
- ◇ Creating a New File
- ◇ Opening an Existing File

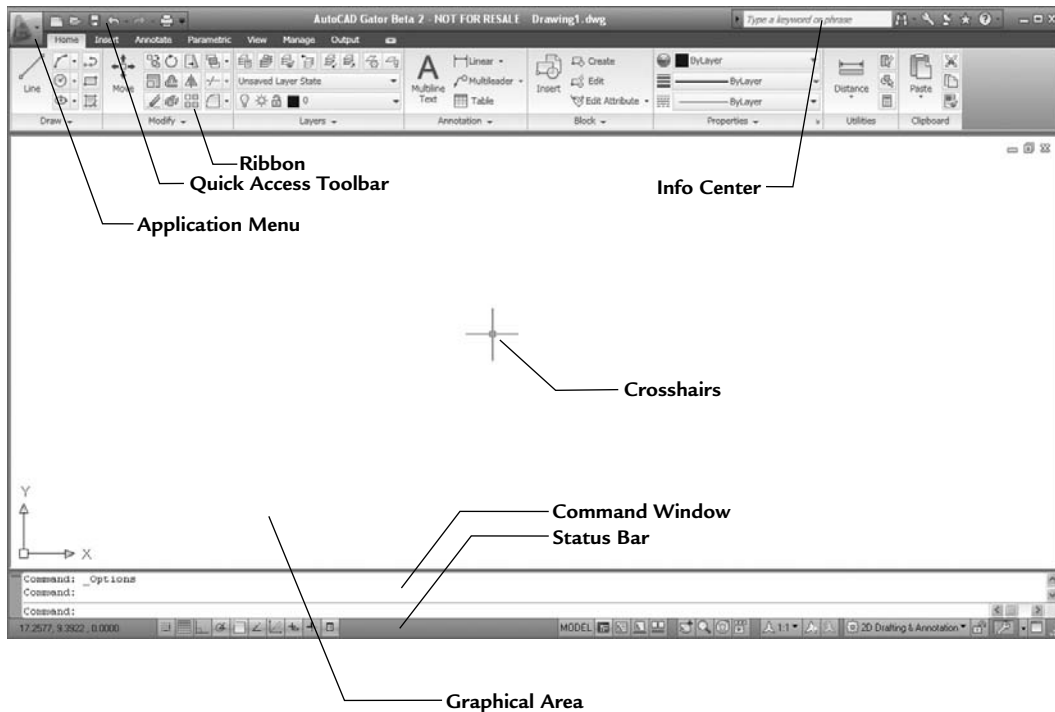
1.1 WHAT IS AUTOCAD®?

- AutoCAD® was one of the first Computer Aided Design/Drafting (CAD) software applications in the world.
- The first version of AutoCAD was released at the end of 1982, and it was designed to be used only on PCs.
- Since 1982, AutoCAD has increased dramatically on a global scale.
- Users can draw both two-dimensional (2D) drawings and three-dimensional (3D) designs in AutoCAD.
- There is another version of AutoCAD called AutoCAD LT that is dedicated to 2D drafting only.

1.2 HOW TO START AUTOCAD® 2010



- There are two ways to start AutoCAD 2010:
 - While installing AutoCAD 2010, the installation program will create a shortcut on your desktop. To start AutoCAD, simply double-click this icon.
 - From the **Windows** taskbar click **Start/All Programs/Autodesk/AutoCAD2010/AutoCAD2010**.
- AutoCAD will start with a new, open drawing file, which will look like the following:



1.3 UNDERSTANDING THE AUTOCAD® 2010 INTERFACE

- The primary methods to reach commands in AutoCAD 2010 will be through **Ribbons** and the **Application Menu**.
- You will use **Ribbons** instead of the normal toolbars.
- This interface will give you more space in the **Graphical Area**, which is your drawing area.

Application Menu

- Click on the **Application Menu**, and you will see the following:

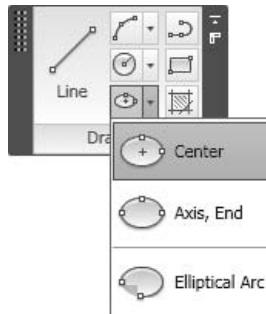


- Using this menu, you can:
 - Create a new file
 - Open an existing file
 - Save the current file
 - Save the current file under a new name (Save As) and/or in a different folder
 - Export the current file to a different file format
 - Print the current file
 - Publish the current file
 - Send the current file to eTransmit or email
 - Use all the functions related to your drawing

- At the lower left-hand corner of the panel you will see a small pushpin. If you click on it, this will be the default view. To return to the previous view, simply click the pushpin again.



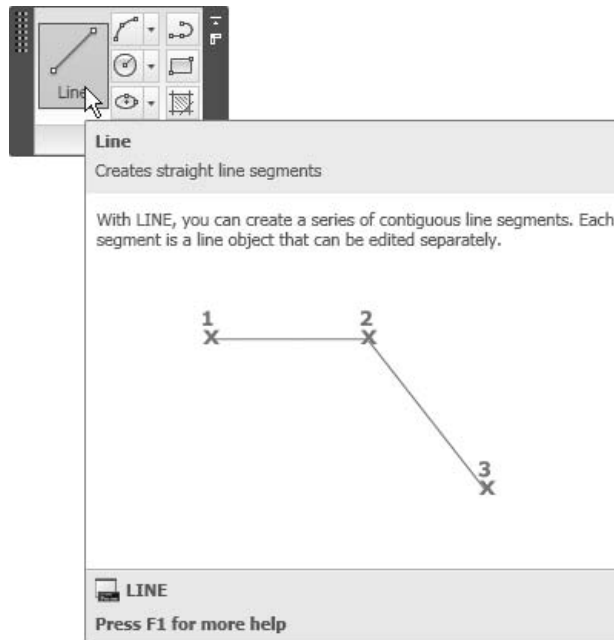
- Some panel buttons may have a small triangle just to the right, which means there are additional options, as shown here:



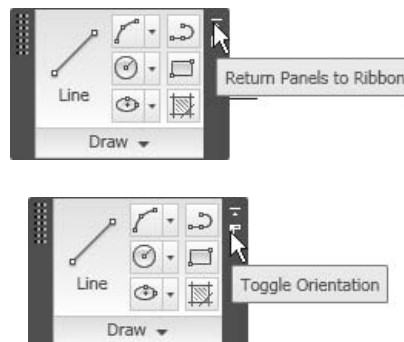
- If you hold your cursor over any button for one second, a small help screen appears:



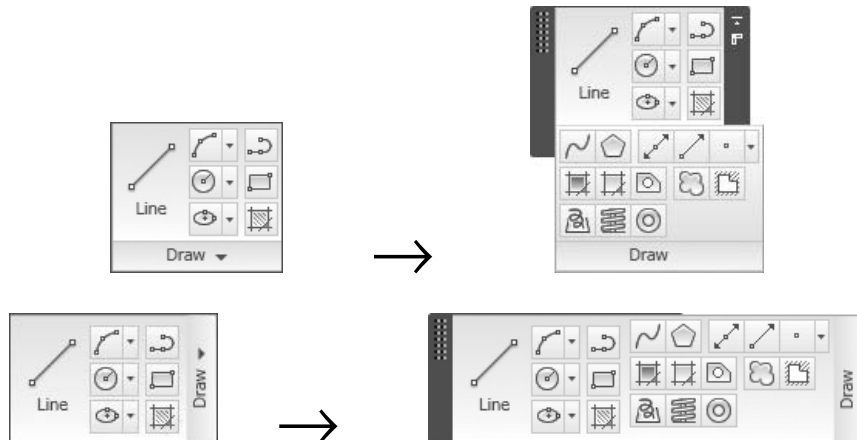
- However, if you hold your cursor over the button for three seconds, you will see an extended help screen:



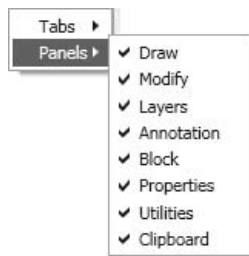
- Panels can be docked or floating. By default, all panels are docked. To make a panel a floating panel, simply click on the name of the panel, hold it, and drag it to its new location.
- If all panels remain docked, you will *not* see any panel outside the tab it belongs to. But if you make any panel a floating panel, then you will be able to see it in all other tabs.
- While the panel is floating, you will see two small buttons on the right-hand side. The following image shows the function of each one:



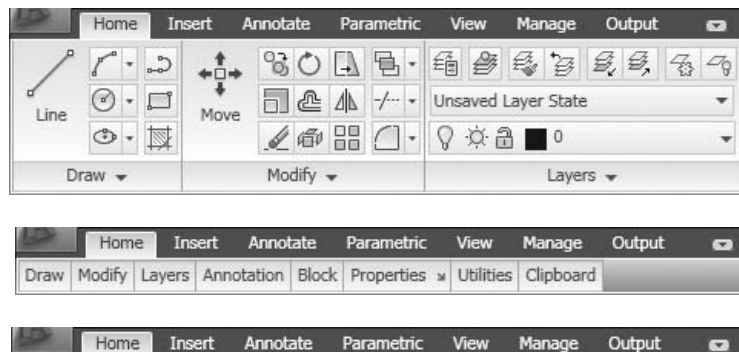
- All panels have two orientations: a vertical orientation and a horizontal orientation. The following illustration shows the tabs and panels that appear for both orientations.



- The number of tabs and panels are predefined by default.
- You can turn off/on any tab/panel by right-clicking on any tab/panel and then selecting the desired tab/panel to be turned on or off.

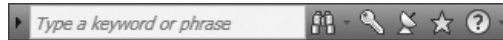


- **Ribbons** have three different shapes. You can alter the shapes by clicking the small arrow at the right:

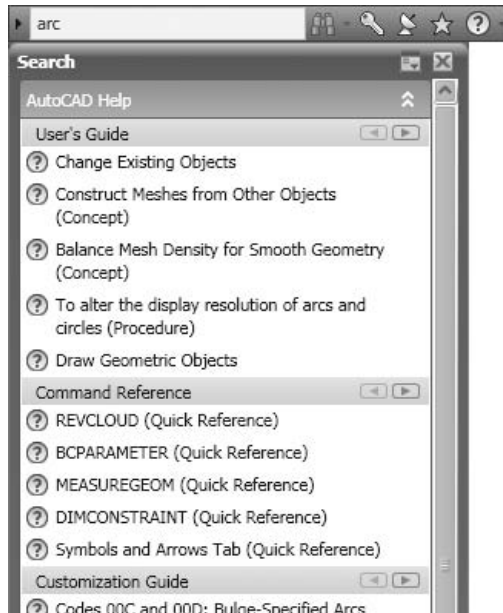


InfoCenter

- At the top-right part of the screen, you will see the **InfoCenter**:



- The **InfoCenter** gives you the opportunity to type in keywords that will enable AutoCAD to search both online and offline resources and provide you with a list of related help topics. See the following example:



- You may still use the conventional **Help** search by clicking the arrow at the right-hand side of the **InfoCenter**:



Command Window

- If you used AutoCAD 25 years ago, the only way to input commands was to use the **Command Window**. You had to memorize all of the AutoCAD commands and type them in. There were no **menus**, **toolbars**, **panels**, or **ribbons**. Although you can still type in the commands, we don't recommend that method.

Graphical Area

- The **Graphical Area** is where you do your actual drawing. You use the crosshairs to specify points in the X,Y plane.
- You can monitor the coordinates of the crosshairs using the left side of the **Status Bar**.

Status Bar

- The **Status Bar** in AutoCAD contains many functions that will help you draft more precisely. We will discuss most of the buttons on the **Status Bar** throughout this book.
- There are two views for the **Status Bar**:
 - By default (Icons)



- Buttons



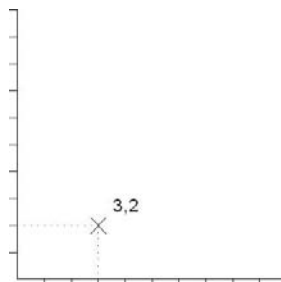
- To switch views, right-click the **Status Bar** to alternate between buttons and icons.



- If you select **Use Icons**, the view will change accordingly.

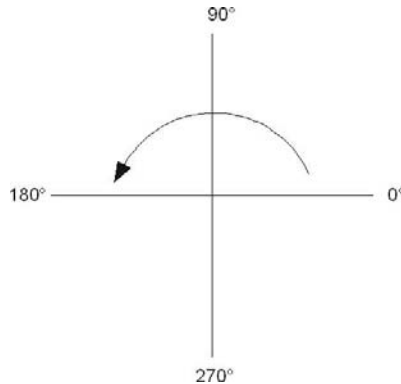
1.4 POINTS IN AUTOCAD®

- Points are defined (and saved) in AutoCAD using the **Cartesian coordinate system**.
- The coordinates will look something like **3.25,5.45**, which is the format of **X,Y**.
- So the first and most traditional way of specifying points in AutoCAD is to type the coordinates whenever you are asked to do so, by typing X,Y (pronounced X comma Y). See the following illustration:



1.5 AUTOCAD® DEFAULT SETTINGS

- Sign convention: positive is up and right.
- Angle convention: positive is counterclockwise (CCW) starting from the east (i.e., 0 angle). See the following illustration:

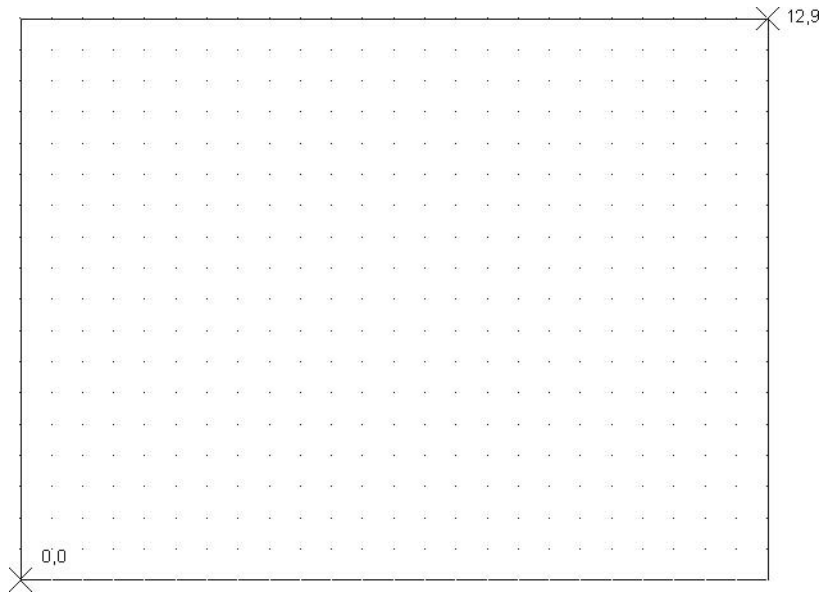


1.6 THINGS YOU SHOULD KNOW ABOUT AUTOCAD®

- The mouse is the primary input device:
 - The left mouse button is always used to select and/or click.
 - The right mouse button, when clicked, offers a drop-down menu.
- The mouse wheel has zooming functions:
 - Zoom in on your drawing by moving the wheel forward.
 - Zoom out of your drawing by moving the wheel backward.
 - Pan (i.e., move through the drawing) by pressing the wheel and holding it and then moving the mouse.
 - Zoom to the edges of your drawing by double-clicking the wheel.
- If you type an AutoCAD command or any input in the **Command Window**, you have to press the [Enter] key to execute it.
- [Enter] = [Spacebar] in AutoCAD.
- To repeat the last AutoCAD command, press [Enter] or [Spacebar].
- To cancel any AutoCAD command, press [Esc].

1.7 DRAWING LIMITS

- AutoCAD offers users an infinite drawing sheet on all sides.
- When you start a new AutoCAD drawing, your viewpoint will be at 0,0,1.
- You are looking at the X,Y plane, using a camera's lens; hence you will see part of your infinite drawing sheet. This part is called the **limits**. See the following:



- In this example, you can see that the **limits** of the drawing are from 0,0 (lower left-hand corner) to 12,9 (upper right-hand corner). This is your working area.
- We will learn how to change **limits** in upcoming chapters.

1.8 UNITS AND SPACES

- One of the vague facts about AutoCAD is that it does not deal with a certain length unit while drafting. Take note of the following points:
 - AutoCAD deals with AutoCAD units.
 - An AutoCAD unit can be anything you want. It can be a meter, centimeter, millimeter, inch, or foot.
 - All of these options are correct as long as you remember your chosen option and stay consistent in both X and Y.
- Also, there are two spaces in AutoCAD, **Model Space** and **Paper Space (Layout)**. You can switch between the two spaces using the **Status Bar**.



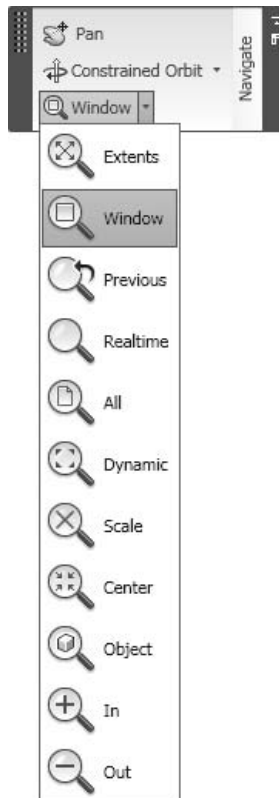
Paper Space (Layout) button

Model Space button

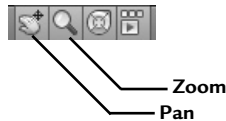
- By default, when you start a new drawing file, you are in **Model Space**.
- In **Model Space** you can create and modify your drawing.
- Once you are ready to make a hardcopy of your drawing file, switch to the **Paper Space (Layouts)** so you can prepare your page setup.
- This is the moment you need to ask, “*What is my unit assumption?*” so that you can properly scale your drawing.
- We will discuss printing in a later chapter.

1.9 VIEWING COMMANDS

- We already discussed using the mouse wheel for zooming in, zooming out, and panning.
- You can also zoom in, zoom out, and pan using the zooming and panning commands.
- If using **Ribbons**, make sure you are in the **View** tab, and, using the **Navigate** panel, select the first button on the left, or the small arrow, to see a list of zooming commands:

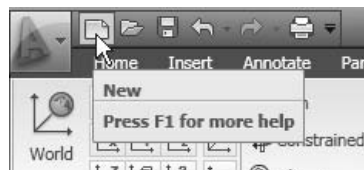


- **Zoom Extents** is used to zoom to all objects.
 - **Zoom Window** is used to specify a rectangle. By specifying two opposite corners, whatever is inside the rectangle will look larger.
 - **Zoom Previous** is used to restore the previous view, up to the last ten views.
 - **Zoom Realtime** is done by clicking the left button on the mouse and holding it. If you move forward, you are zooming in; if you move backward, you are zooming out.
 - **Zoom Dynamic** is used with the **Zoom Window** first. You will see the whole drawing and your current place (shown as a dotted green line), go to the new location, and press [Enter].
 - **Zoom Scale** is used to input a scale factor. If you type in a number less than 1, you will see the drawing smaller. If the scale factor you type in is greater than 1, you will see the drawing larger. If you put the letter x after the number (e.g., 2x) the scale will be relative to the current view.
 - **Zoom Center** is used to specify a new center point for the zooming, along with a new height.
 - **Zoom Object** is used to zoom to certain selected objects. AutoCAD will ask you to select objects and the selected objects will fill the screen.
 - **Zoom In** is not really a zoom option, but rather a programmed option equal to the **Zoom Scale** with a scale factor of 2x.
 - **Zoom Out** is just like **Zoom In**, but with a zoom factor of 0.5x.
- Also on the Status Bar, you can use the following two buttons:

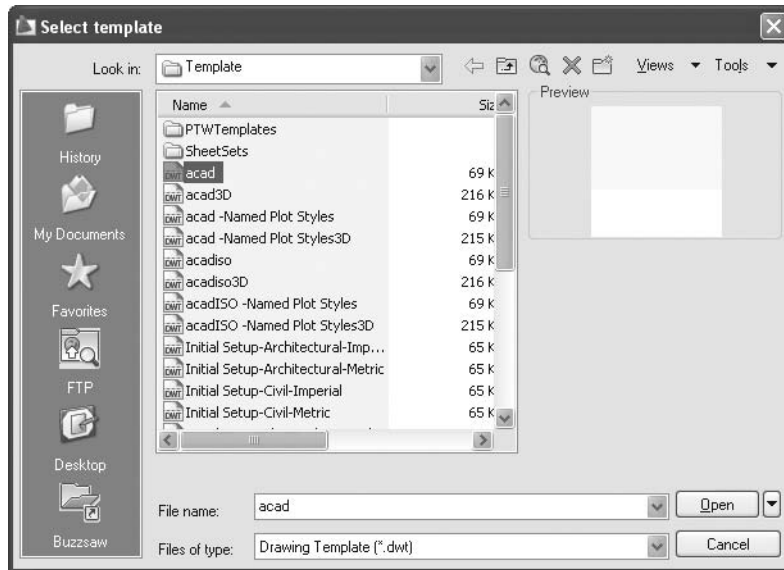


1.10 CREATING A NEW FILE

- To create a new file based on a premade template, click on the **New** button on the **Quick Access Toolbar**:



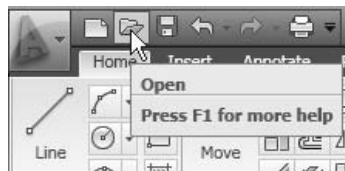
- The following dialog box will appear:



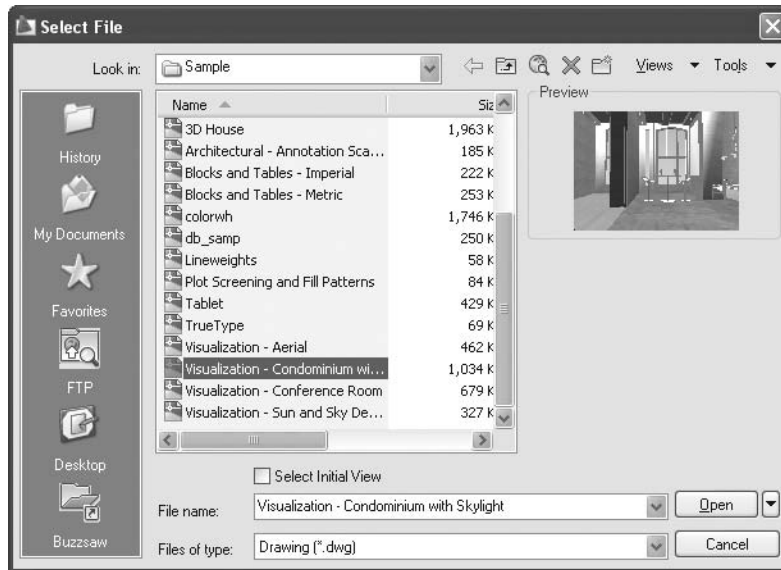
- This dialogue box will allow you to select the desired template.
- AutoCAD template files have the extension **.dwt*.
- AutoCAD 2010 has lots of premade templates you can use, or you can create your own template.
- For now, we will use *acad.dwt* for some of our exercises.
- Click **Open** to start a new file.

1.11 OPENING AN EXISTING FILE

- To open an existing file for further editing, click the **Open** button from the **Quick Access Toolbar**.



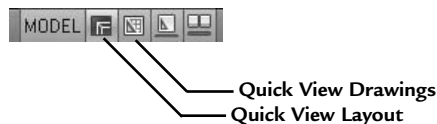
- The following dialog box will appear:



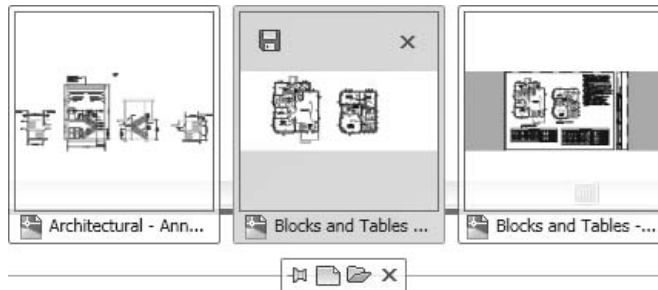
- Specify the hard drive, and the folder your file resides in.
- AutoCAD drawing files have the extension **.dwg*.
- If you want to open a single file, select the file and click open (you can also double-click on the file's name).
- If you want to open more than one file, select the first file name, then hold the [Ctrl] key on the keyboard, and click the other file names.
- You can open as many files as you wish.
- When you are done, click **Open**.

Quick View

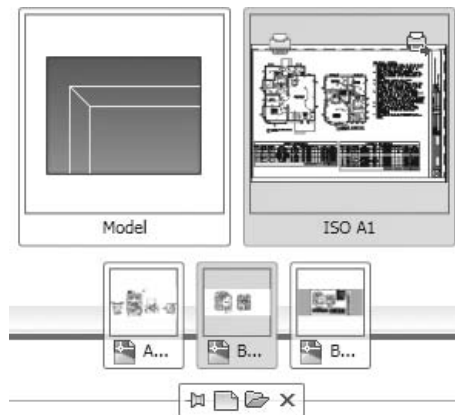
- If you open more than one file, you can use two functions in the **Status Bar**: **Quick View Drawings** and **Quick View Layouts**.



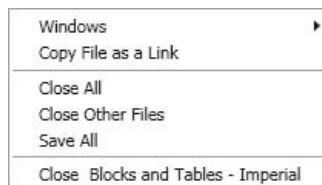
- If these two buttons are switched on, you will see the following:



- You will see all of the open files.
- In order to jump from one file to another, click the window of the desired file.
- When you hover over any of the files, you will see the layouts of the file, and the picture will change to:



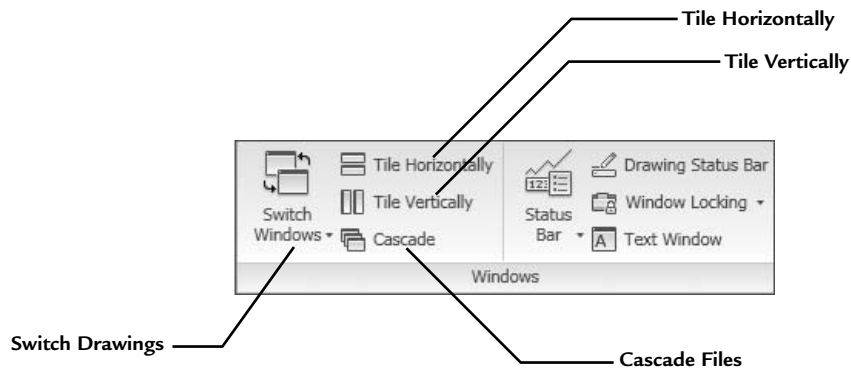
- Also, you will see a small toolbar at the bottom of the screen that will enable you to:
 - Close the **Quick View**
 - Open a file
 - Start a new file
 - Pin **Quick View Drawings**
- If you right-click the **Quick View** button of any file, the following menu will appear:



- You can close all files
- Close other files except the current file
- Save all files
- Close the current file

Organizing Files

- Make sure you are in the **View** tab on the **Ribbon**. There are several ways to organize the files by using the **Windows** panel.



- The **Switch Drawings** button will show you a list of the opened files. The current file will be listed with a checkmark (✓). If you want another file, select the name of the desired file.
- The rest of file commands, such as **Save**, **Save As**, and **Exit**, are identical to those in other Windows applications.



INTRODUCING AUTOCAD® 2010



Exercise 1

1. Start AutoCAD 2010.
2. From the **Quick Access Toolbar**, click the **Open** button.
3. Go to the **Samples** folder under the AutoCAD 2010 folder. Open the following three files:
 - a. Blocks and Tables—*Metric.dwg*
 - b. *dbsamp.dwg*
 - c. Architectural and Annotation Scaling—*Multileader.dwg*
4. Using **Quick View Drawings**, jump from one file to another. Using **Quick View Layouts**, take a look at the layouts in each file.
5. Using one of the open files, use the mouse wheel—like any of the other methods discussed in this chapter—and zoom in, zoom out, and pan.
6. Using the right-click menu on the **Quick View Drawings**, close all files without saving.

CHAPTER REVIEW

1. You can close all open files using one command:
 - a. True
 - b. False
2. CAD stands for_____.
3. In AutoCAD there are two available spaces: **Model Space** and **Paper Space**. Which of the following statements is true?
 - a. You draw on **Model Space** and print from **Paper Space**.
 - b. You draw in **Paper Space** and print from **Model Space**.
 - c. There is only one space in AutoCAD.
 - d. **Model Space** is only for 3D design.
4. Positive angles start from the north.
 - a. True
 - b. False
5. AutoCAD is one of the few software applications that allows users to:
 - a. Connect to the Internet.
 - b. Type commands using the keyboard.
 - c. Accept Cartesian coordinates.
 - d. Create positive angles that are CCW.
6. _____ is a tool in AutoCAD that allows users to see all open files in small windows.

CHAPTER REVIEW ANSWERS

1. a
2. Computer Aided Design/Drafting
3. a
4. b
5. b
6. **Quick View Drawings**

Chapter 2 DRAFTING USING AutoCAD® 2010

In This Chapter

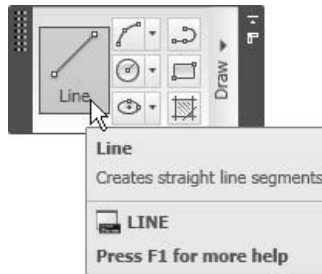
- ◇ Introduction
- ◇ The **Line** Command
- ◇ Drafting Using **Dynamic Input**
- ◇ Precision Method 1: **Snap** and **Grid**
- ◇ Precision Method 2: **Direct Distance Entry** and **Ortho**
- ◇ The **Arc** Command
- ◇ The **Circle** Command
- ◇ Precision Method 3: **Object Snap (OSNAP)**
- ◇ **Object Snap Tracking (OTRACK)**
- ◇ The **Pline** Command
- ◇ **Polar Tracking**
- ◇ The **Erase** Command
- ◇ **Oops, Undo, and Redo** Commands
- ◇ **Redraw** and **Regen** Commands

2.1 INTRODUCTION

- The two most important things in drafting are:
 - Precision
 - Speed
- You always want to finish your drawing as fast as possible, yet you do not want to undermine your drawing's precision.
- It is best to learn precision before speed because it is easier to learn to speed up the creation process than it is to improve accuracy.
- In this chapter we will tackle many commands, but learning to draft with precision is most important.

2.2 THE LINE COMMAND

- The **Line** command is used to draw segments of straight lines.
- There are many methods available to draw precise shapes using the **Line** command (which we will learn later). For now, we will type coordinates in the **Command Window**.
- On the **Ribbon**, make sure you are in the **Home** tab. Using the **Draw** panel, click the **Line** button.



- The following prompts will appear:

Specify first point: *(type in the coordinate of the first point)*

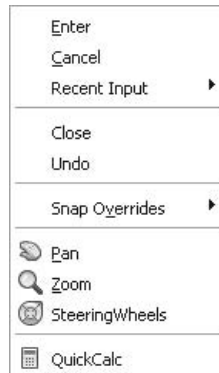
Specify next point or [Undo]: *(type in the coordinate of the second point)*

Specify next point or [Undo]: *(type in the coordinate of the third point)*

Specify next point or [Close/Undo]: *(type in the coordinate of the fourth point)*



- You can use the **Undo** option at any time to undo the last specified point, hence the last specified segment.
- After you draw two segments, the **Close** option will be available to connect the last point to the first point and to end the command.
- Other ways of ending the command include pressing [Enter] or [Spacebar].
- You can also press [Esc] to end the **Line** command.
- If you are using the **Line** command and you right-click, you will get the following menu (which is identical to the command prompt):

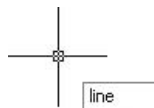


2.3 DRAFTING USING DYNAMIC INPUT

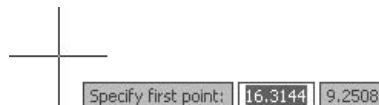
- By default, the **Dynamic Input** is turned on, so anything you type in the **Command Window** will appear on the screen beside the AutoCAD® cursor.
- Make sure the **Dynamic Input** button is turned on:



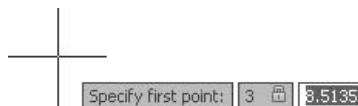
- For example, if you type the word “line,” here is how it will look on the screen:



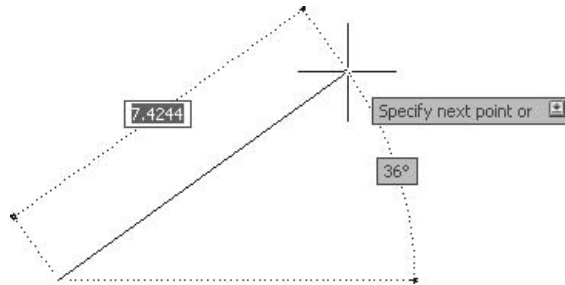
- When you press [Enter], the following will appear:



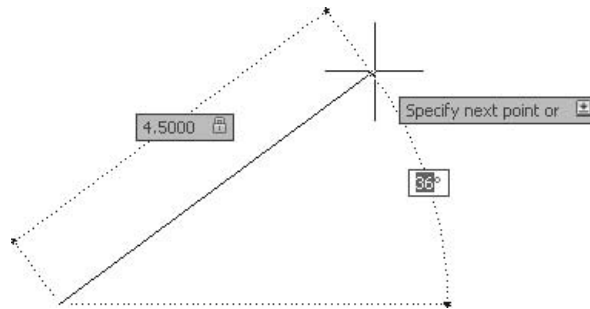
- Type the X coordinates, then press the [Tab] key and you will see the following:



- Type the Y coordinates, then press [Enter] and **Dynamic Input** will show the length and the angle of the line to be drawn (the angle is measured from the east and incremented by 1 degree).



- Specify the length, press the [Tab] key, then type the angle and press [Enter]:

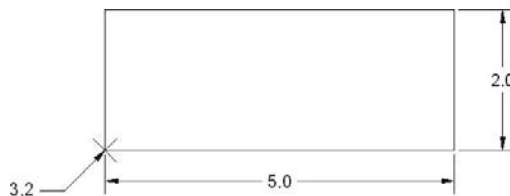


DRAWING LINES: FIRST METHOD



Exercise 2

1. Start AutoCAD 2010.
2. Open the file *Exercise_02.dwg*.
3. Make sure that **Polar Input** is off and **Dynamic Input** is on.
4. Draw the following lines using the **Line** command and **Dynamic Input**:



5. Save the file and close it.



- Start the **Line** command and then type the coordinates of the first point. While **Dynamic Input** is on, specify the length of the line, press [Tab], and then specify the angle. Do the same for the other line segments.

2.4 PRECISION METHOD 1: SNAP AND GRID

- As you can see, the only method we used to precisely specify points in the X,Y plane was to type the coordinates using **Dynamic Input**.
- We do this because we cannot depend on the mouse to specify precise points.
- In order to use the mouse precisely, we have to use tools to control its movement.
- Snap** is the only tool in AutoCAD that can help us control the movement of the mouse.
- Using the **Status Bar**, click on the **Snap Mode** button.



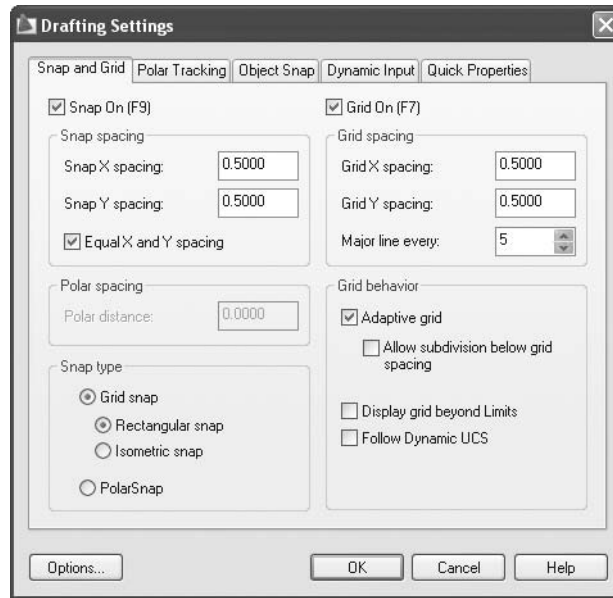
- Now, move to the **Graphical Area** and watch the mouse jump to exact points.
- The **Grid** will show a grid of points on the screen similar to grid paper, which is used in drawing diagrams (these points are not real points).
- A **Grid** on its own is not accurate, but it is a helpful tool to use with **Snap**.
- On the **Status Bar**, click on the **Grid** button.



- You can now see the points displayed on the screen.
- If you think the default values for either **Snap** or **Grid** do not satisfy your needs, simply right-click one of the two buttons and the following shortcut menu will appear:



- Select **Settings** and the following dialog box will appear:



- By default, **Snap X spacing** and **Snap Y spacing** are equal; **Grid X spacing** and **Grid Y spacing** are also equal. If you want this to continue, make sure that the checkbox **Equal X and Y spacing** is always checked.
- By default, if you are working with 2D you will only see **Grid** dots. If you work with 3D, you will see **Grid** lines; therefore, you must set the **Major line** spacing.
- Also, all of the settings of **Grid behavior** are meant for 3D drawings.
- Make sure that **Snap type** is **Grid snap** (we will discuss **Polar Snap** shortly). If you are creating a 2D drawing, then select the **Rectangular snap** option. If you are creating a 3D drawing, select **Isometric snap**.
- If you want **Grid** to follow **Snap**, set the two grid values to zero.
- You can use function keys to turn on both **Snap** and **Grid**:
 - F9 = **Snap** on/off
 - F7 = **Grid** on/off



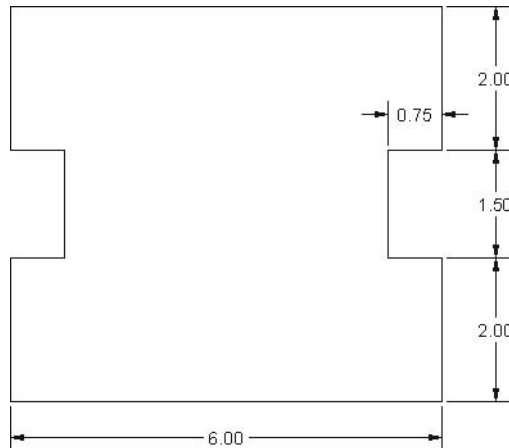
SNAP AND GRID



Exercise 3

1. Start AutoCAD 2010.
2. Open the file *Exercise_03.dwg*.

3. Using **Snap** and **Grid**, draw the following lines without typing any coordinates on the keyboard. Start from the lower left-hand corner with 2.75,2.25 (do not draw the dimensions):



4. Save the file and close it.

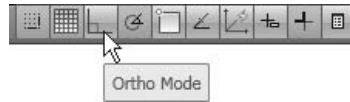


- Change the **Snap X spacing** to 0.25 first and set the grid values to 0. Switch both **Snap** and **Grid** on and draw the lines as required.

2.5 PRECISION METHOD 2: DIRECT DISTANCE ENTRY AND ORTHO

- Because we know that lines in AutoCAD are **vectors**, we need to specify a length and an angle to successfully draw them.
- **Ortho** is a tool that will force the cursor to always give us orthogonal angles (i.e., 0, 90, 180, and 270).
- **Direct Distance Entry** is a very handy tool in drafting; if the mouse is already directed toward an angle, just type in the distance and press [Enter].
- Combining the two tools will allow us to draw lines with precise lengths and angles.
- Follow these steps:

- On the **Status Bar**, click the **Ortho Mode** button.



- Start the **Line** command.
- Specify the first point.
- Move the mouse to the right, up, left, and down. Notice how it only gives you orthogonal angles.
- Use the desired angle, type in the distance, and press [Enter].
- Continue with other segments using the same method.



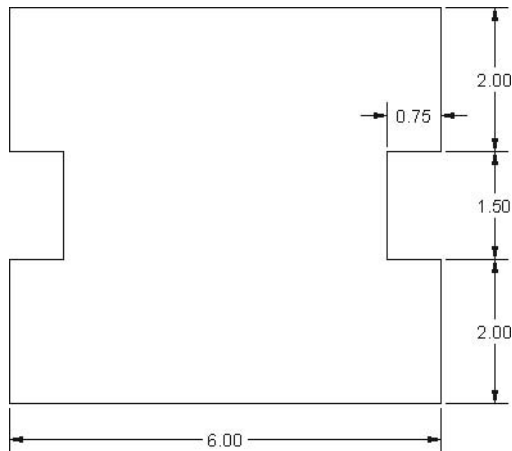
- You can also use **Direct Distance Entry** with **Dynamic Input**.

DIRECT DISTANCE ENTRY AND ORTHO



Exercise 4

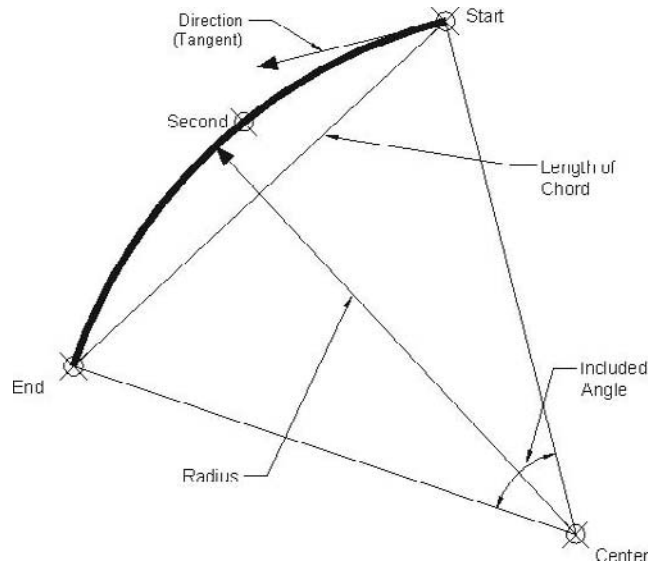
1. Start AutoCAD 2010.
2. Open file *Exercise_04.dwg*.
3. Using **Ortho** and **Direct Distance Entry**, draw the shape that follows starting from the lower left-hand corner with 3,2 (without dimensions):



4. Save the file and close it.

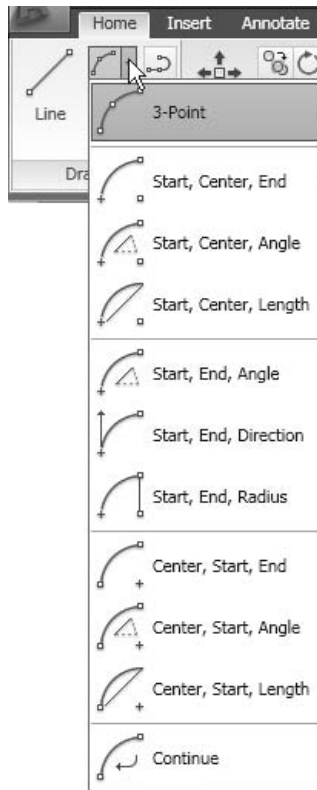
2.6 THE ARC COMMAND

- The **Arc** command is used to draw circular arcs (the arc part of a circle).
- Take a look at the following illustration:



- The information that AutoCAD recognizes about an arc includes:
 - Start point
 - Second point (not necessarily the midpoint)
 - Endpoint
 - Center point
 - Radius
 - Length of chord
 - Included angle (angle between **Start**, **Center**, **End**)
 - Direction (the tangent passes through the start point)
- AutoCAD only needs three pieces of information to draw an arc, but not just any three.
- AutoCAD will start asking you to make your first input, choosing either the start point or the center point, and based on that choice it will ask you to specify the second piece of information, and so on.

- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Draw** panel, click the **Arc** button (the small arrow at the right). You will see the following:



- Before you start, specify the desired method from the menu and AutoCAD will take it from there.
- Always think counterclockwise (CCW) when specifying points.

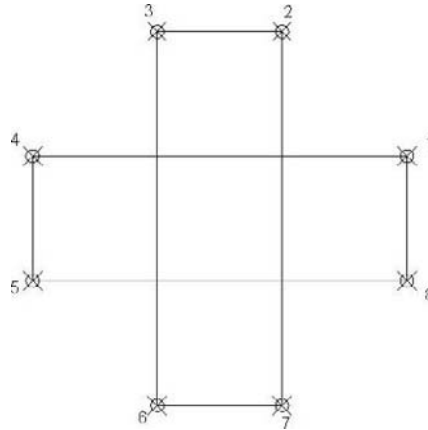


DRAWING ARCS

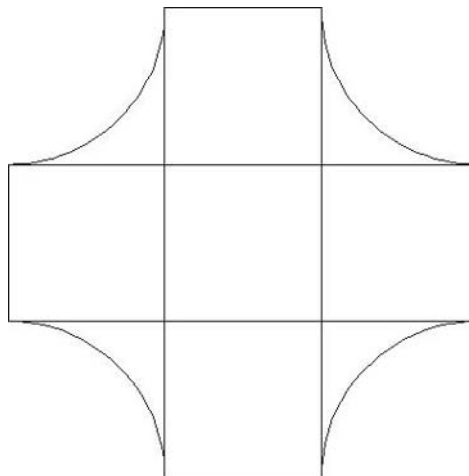


Exercise 5

1. Start AutoCAD 2010.
2. Open the file *Exercise_05.dwg*. You will see the following shape:



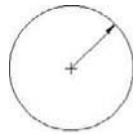
3. Turn on **Snap** and **Grid**.
4. Draw the first arc between point (1) and (2) using **Start, End, Angle**, where point (1) is the start point and the angle = -90 .
5. Draw the second arc between point (3) and (4) using **Start, End, Direction**, where point (3) is the start point and the direction = 270 .
6. Draw the third arc between point (5) and (6) using **Start, Center, End**, where point (6) is the start point and the point at the lower left is the center point (you can specify it using **Snap** and **Grid**).
7. Draw the fourth arc between point (7) and (8) using **Start, Center, Length**, where point (8) is the start point, the point at the lower right is the center point, and the length of the chord is the distance between (8) and (7).
8. The shape should look like this:



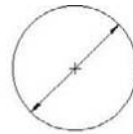
9. Save the file and close it.

2.7 THE CIRCLE COMMAND

- The **Circle** command is used to draw a circle.
- There are six different methods available to draw a circle in AutoCAD.
- To use the first two methods you have to know the center of the circle. They are:

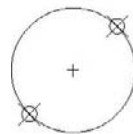


Center / Radius

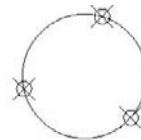


Center / Diameter

- To use the third method, select three points opposite each other (see the following illustration) on the perimeter of the circle.
- To use the fourth method, specify two points opposite each other on the perimeter of the circle. The distance between them is equal to the diameter.

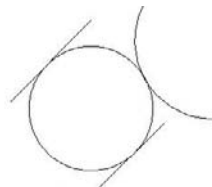


2 Points

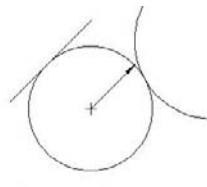


3 Points

- To use the fifth method, you should have two objects already drawn—we can consider them as tangents—then specify a radius.
- To use the sixth method, you should specify three tangents by selecting three objects.

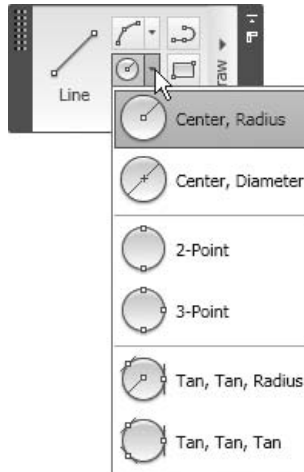


Tangent/Tangent/Tangent



Tangent/Tangent/Radius

- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Draw** panel, click the **Circle** button (the small arrow at the right). You will see the following:



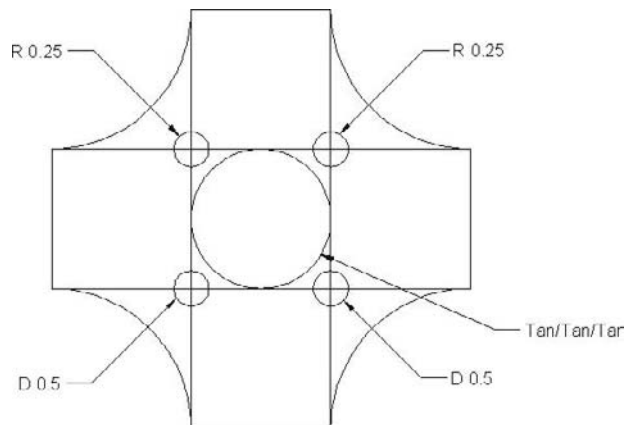
- Before you start, specify the desired method from the menu and AutoCAD will take it from there.

DRAWING A CIRCLE



Exercise 6

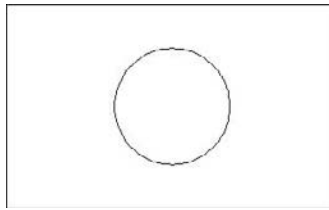
1. Start AutoCAD 2010.
2. Open the file *Exercise_06.dwg*.
3. Make sure that **Snap** and **Grid** are on.
4. Draw the five circles, which should look like the following:



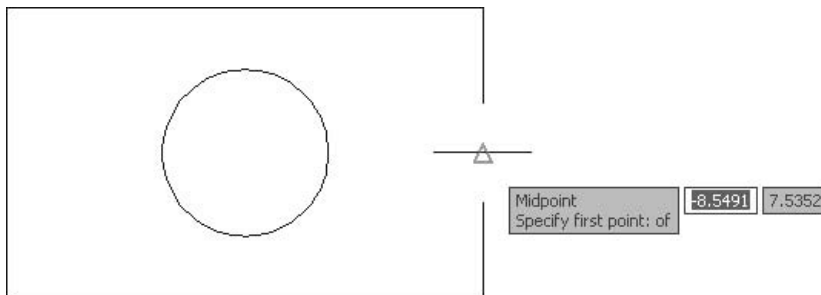
5. Save the file and close it.

2.8 PRECISION METHOD 3: OBJECT SNAP (OSNAP)

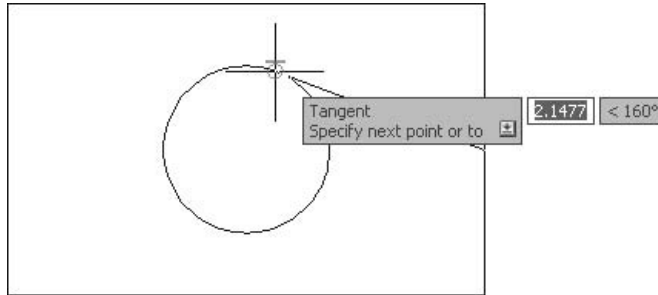
- AutoCAD keeps a full record of each object in each drawing.
- **Object Snap (OSNAP)** is a tool that helps you utilize these records when you need to specify points on objects already precisely drawn without knowing the points.
- For example,
 - Assume we have the following shape:



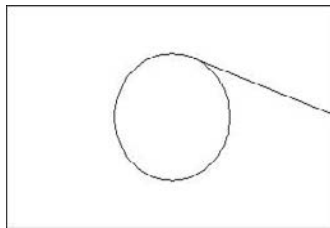
- We have no information about any object in this shape.
- We were asked to draw a precise line from the midpoint of the right line to the tangent to the top of the circle.
- The command to draw is **Line**. AutoCAD asked us to specify the first point and we typed **mid** and pressed [Enter] (or [Spacebar]), then went directly to the upper line and a yellow triangle appeared. We clicked:



- AutoCAD then asked us to specify the next point. We typed **tan** and pressed [Enter] (or [Spacebar]), then went directly to the top of the circle and a yellow circle appeared. We clicked it and then pressed [Enter] to end the command:



- Mission accomplished.



- Some of the **OSNAPs** are used to catch the following:
 - The endpoint of an object
 - The midpoint of an object
 - The intersection of two objects
 - The center of an arc or a circle
 - The quadrant of an arc or a circle
 - The tangent of an arc or a circle
 - The perpendicular point on an object
 - A point on an object nearest to your click point
- We will discuss more **OSNAPs** as we learn additional commands.
- There are three ways to use the **OSNAPs** whenever you are asked to specify a point, these include: Typing, [Shift] + right-click, Running **OSNAP**.

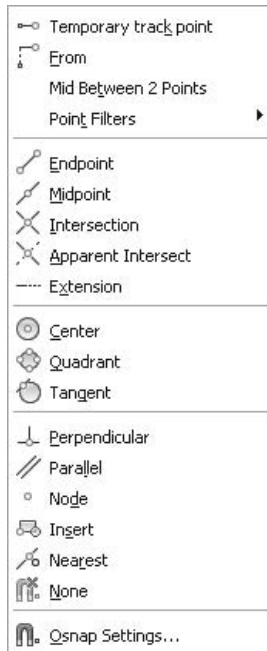
NOTE

Typing

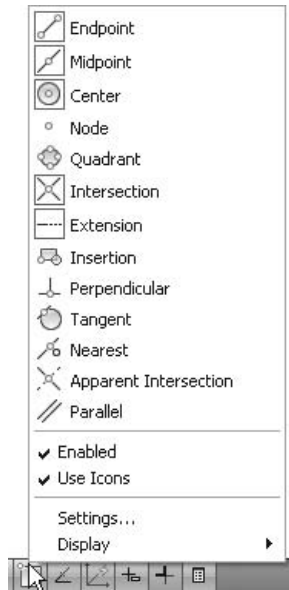
- Type the first three letters of the desired **OSNAP** such as, **end**, **mid**, **cen**, **qua**, **int**, **per**, **tan**, and **nea**. This is a very old method, but often used among seasoned users of AutoCAD.

[Shift] + Right-click

- Hold the [Shift] key and right-click. The following menu will appear. Select the desired **OSNAP**.

**Running OSNAP**

- This method is the most practical method of those we have mentioned.
- You will select an **OSNAP** and it will run all the time. The next time you are asked to specify an endpoint, for example, simply go to the desired point and it will be acquired immediately.
- There are two ways to activate running **OSNAP**:
 - On the **Status Bar**, right-click the **OSNAP** button and the following menu will appear:



- In this example, **Endpoint**, **Midpoint**, **Center**, **Intersection**, and **Extension**, are all running (each icon has a frame around it).
- You can also select **Settings** and the following dialog box will appear:



- Switch on the desired **OSNAP** and click **OK**.

2.9 OBJECT SNAP TRACKING (OTRACK)

- If you have a rectangle, and you want to draw a circle where its center will coincide with the exact center of the rectangle, **Object Snap Tracking (OTRACK)** will help you do this without drawing any new objects to facilitate specifying the exact points.
- **OTRACK** uses **OSNAP**s of existing objects to steal the coordinates of the new point.
- On the **Status Bar**, click the **OTRACK** button.



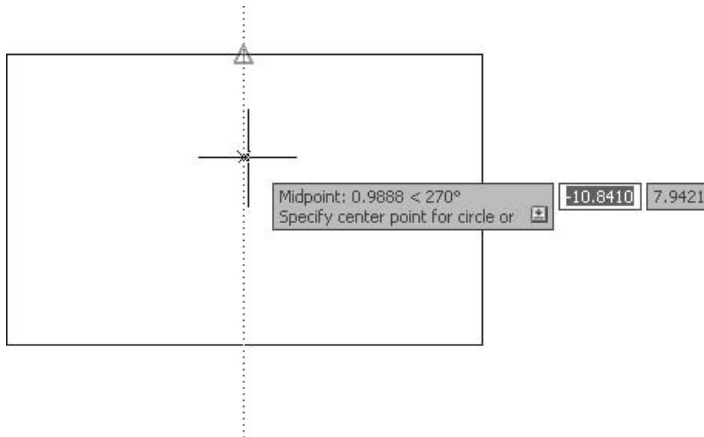
- Make sure that **OSNAP** is also on, as **OTRACK** alone would not do anything.

Example of Two-points **OTRACK**

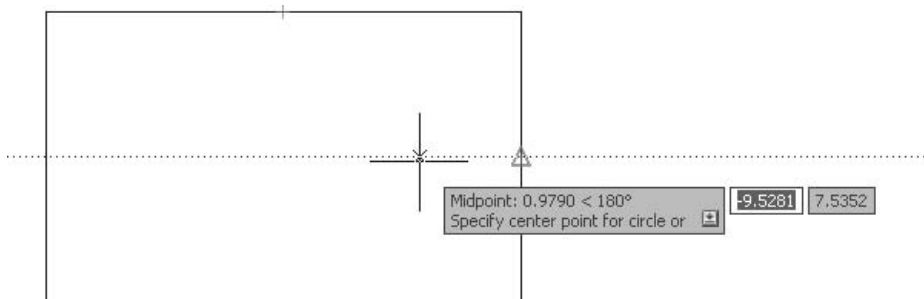
- Let's look at an example where we will use two points to specify one point.
 - Assume we have the following rectangle:



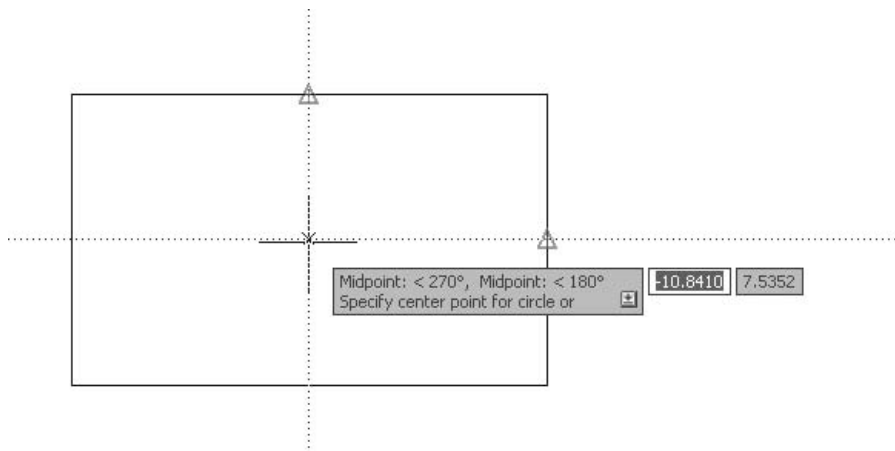
- Make sure that **OSNAP** and **OTRACK** are both turned on. Make sure that **Midpoint** in **OSNAP** is also turned on.
- Start the **Circle** command, which will ask you to specify the center point.
- Go to the upper (or lower) horizontal line and move to the midpoint and hover for a couple of seconds, then move up or down. You will see an infinite line extending in both directions (do not click), just like the following:



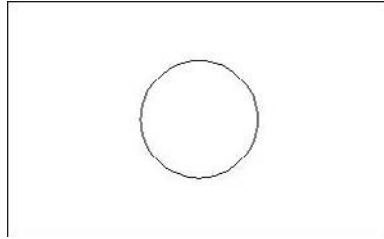
- Go to the right (or left) vertical line and move to the midpoint and hover for couple of seconds, then move right or left. You will see an infinite line extending in both directions, just like the following:



- Now, move your cursor to where you think the two infinite lines should intersect:

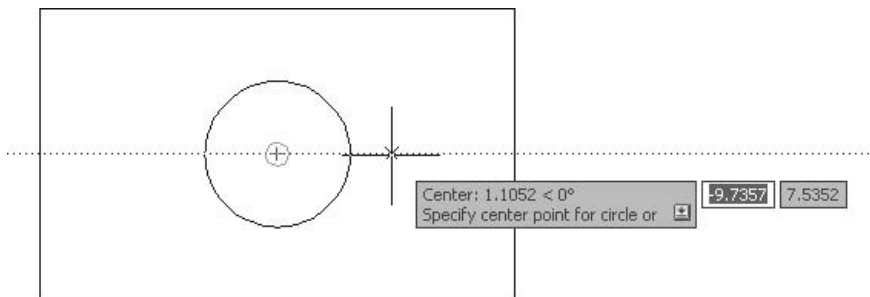


- Once you see the two infinite lines, click. You have specified the center point of the circle. You can now type in the radius (diameter) of the circle:

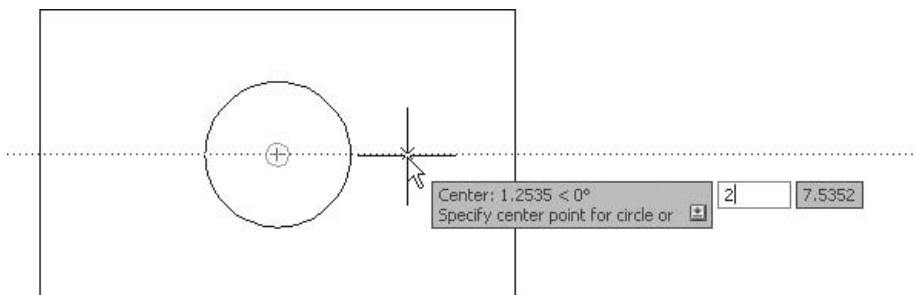


Example of One-point **OTRACK**

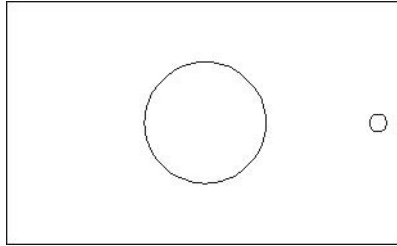
- Let's look at another example. This time we will use one point to specify one point:
 - Continue with the same rectangle used in the last example.
 - Start the **Circle** command, which will ask you to specify the center point.
 - Make sure both **OSNAP** and **OTRACK** are turned on, and also turn on **Center** in **OSNAP**.
 - Go to the center point of the existing circle, and hover for couple of seconds, then move to the right. An infinite line will appear:



- Type 2 (or any distance) and press [Enter]:



- The center of the new circle will be specified, then type in the radius. You will see the following:



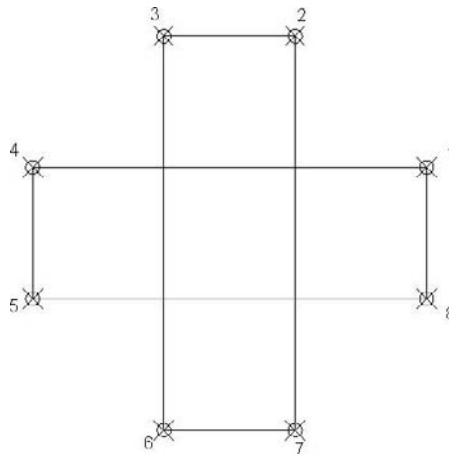
- If you hover over a point for couple of seconds to produce the infinite line and then discover that it is not the desired point, simply go to the same point again, and hover over it for a couple of seconds to disable it.

DRAWING USING OSNAP AND OTRACK



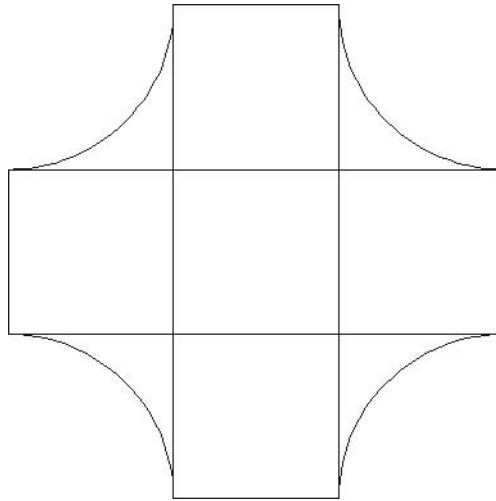
Exercise 7

1. Start AutoCAD 2010.
2. Open the file *Exercise_07.dwg*.

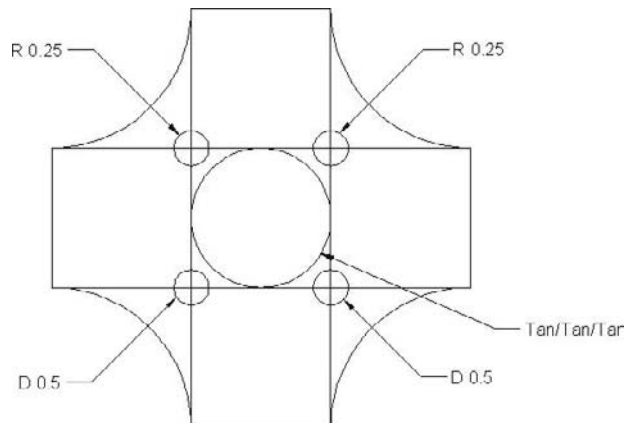


3. Make sure that **Snap** and **Grid** are both turned off.
4. Using **OSNAP**, switch on **Endpoint**.
5. Draw the first arc between points (1) and (2) using **Start, End, Angle**, where point (1) is the start point and the angle = -90 .
6. Draw the second arc between points (3) and (4) using **Start, End, Direction**, where point (3) is the start point and the direction = 270 .

7. Draw the third arc between points (5) and (6) using **Start, Center, End**, where point (6) is the start point. To specify the center point, use **OTRACK** between points (5) and (6).
8. Draw the fourth arc between points (7) and (8) using **Start, Center, Length**, where point (8) is the start point. To specify the center point, use **OTRACK** between points (7) and (8), where the length of the chord is the distance between (8) and (7).
9. The shape should look like the following:



10. Using **OSNAP**, turn **Intersection** on. Use **Intersection** to specify the center points for the five small circles. (By default, AutoCAD will activate **Tangent** when you use the **Tan, Tan, Tan** method.)



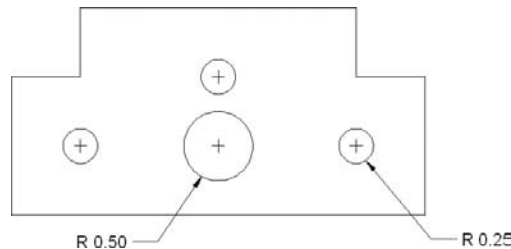
11. Save the file and close it.

DRAWING USING OSNAP AND OTRACK



Exercise 8

1. Start AutoCAD 2010.
2. Open the file *Exercise_08.dwg*.
3. Turn **OSNAP** on and set the following: **Endpoint**, **Midpoint**, and **Center**.
4. Turn on **OTRACK**.
5. Draw the four circles and specify the center using **OSNAP** and **OTRACK**.



6. Save the file and close it.

2.10 THE PLINE COMMAND

- Pline means “polyline” and poly means many—so, if you exchange many with poly, the new name would be “many lines.”
- To begin, let’s compare the **Line** command and the **Pline** command.

Line

Each segment is an object

Lines only

No width

Pline

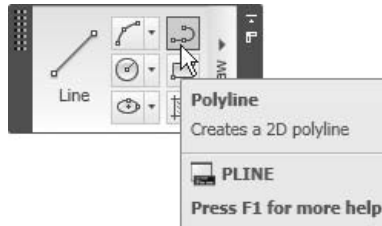
One object with vertices

Lines and arcs

Variable width for start and finish



- As you can see from the comparison, there are mainly three differences between the two commands.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Draw** panel, click the **Polyline** button:



- The following prompt will appear:

Specify start point:

Current line-width is 0.9000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]:

- After you specify the first point, the **Pline** command will give you the current polyline width (in our example it is 0.90), then it will ask you to specify the next point. You can use all of the methods we learned in the **Line** command.
- If you do not want to specify the second point, you can choose from the following: **Arc**, **Halfwidth**, **Length**, and **Width**.

Arc

- By default, the **Pline** command will draw lines.
- You can change the mode to draw arcs by selecting the **Arc** option. The following prompt will appear:

Specify endpoint of arc or

[Angle/Center/Close/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:

- We learned in the **Arc** command that AutoCAD needs three pieces of information to draw an arc.
- AutoCAD already knows the start point of the arc, which is the start point of the polyline or the endpoint of the last line segment.
- AutoCAD will assume that the direction of the arc will be the same angle as the last line segment. Users have the right to accept or reject this assumption.
- If you accept this assumption, then AutoCAD will ask you to specify the endpoint of the arc.
- If you reject this assumption, then you will need to specify the second piece of information from the following:

- Angle of arc, then center or radius
- Center, then angle or length
- Direction, then end
- Radius, then end or angle
- Second, then end

Halfwidth

- The first method is to specify the width of the polyline.
- Specify the halfwidth of the polyline from the center to one of its edges, something like the following:



- When you select this option, AutoCAD will give you the following prompt:
Specify starting half-width <1.0000>:
Specify ending half-width <1.0000>:
- In this example, the halfwidth was 1.0 for both the start point and endpoint.

Length

- In the **Pline** command, if you draw an arc, then switch to the **Line** command to draw a line segment, and if you want the line to be tangent to the arc, then select this option.
- This option will assume the angle to be the same of the last segment, hence, you will only be asked for the length. The following prompt will appear:

Specify length of line:

Width

- Width is the same as halfwidth, but instead, you have to input the full width. See the following illustration:



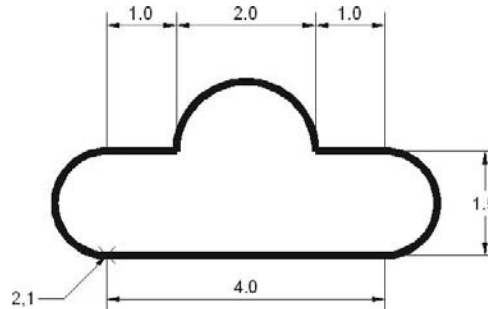
- The **Undo** and **Close** options are the same options as in the **Line** command.
- If you choose to close in the **Arc** option, it will close the shape by an arc.

DRAWING POLYLINES



Exercise 9

1. Start AutoCAD 2010.
2. Open the file *Exercise_09.dwg*.
3. Using **Ortho** and **Direct Distance Entry**, draw the following shape (without dimensions) using the **Pline** command with width = 0.1.



4. Save the file and close it.



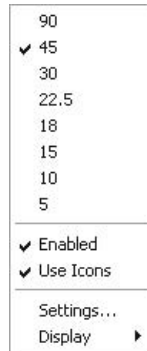
- Take note of the following tips:
 - In order to draw the large arc, use angle = 180.
 - Before you draw the last arc, change the mode to **Arc** and select **Close**.

2.11 POLAR TRACKING

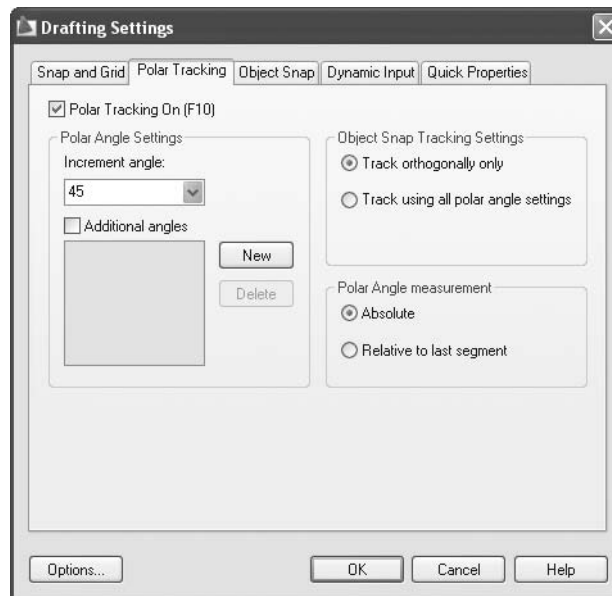
- We learned that we can force the cursor to give us four orthogonal angles (0, 90, 180, 270) using **ORTHO**.
- If we want other angles, such as 30 and its multiples or 60 and its multiples, **ORTHO** would not help us. For this reason, AutoCAD provides another powerful tool called **Polar Tracking**.
- Polar Tracking** allows you to have rays starting from your current point pointing toward angles such as 30, 60, 90, 120, and so on. You can use **Direct Distance Entry** just as we did with **ORTHO**.
- On the **Status Bar**, click the **Polar Tracking** button.



- In order to select the desired angle, right click on the button and you will see the following menu:



- Select the desired angle and you will get it, along with its multiples.
- If you want more control, right-click on the button and select **Settings**. The following dialog box will appear:



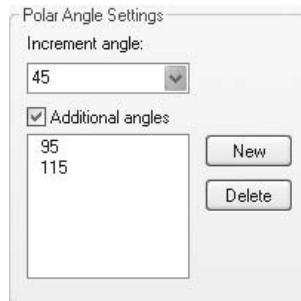
Increment Angle

- Under **Polar Angle Settings**, select the **Increment angle** (a list in a drop-down menu where you will find predefined angles). Select the desired angle.

- If the desired angle is not on the list, simply type your own angle.
- Based on this example, users will have rays in angles 0, 30, 60, 90, 120, etc.

Additional Angles

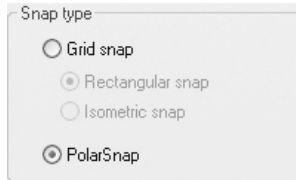
- Sometimes, in the design process, you will need some odd angles that the **Increment angle** option cannot give you, such as 95 or 115.
- The **Additional angles** option will help you set these odd angles.
- Using the same dialog box, check the **Additional angles** box.
- Click the **New** button and type in the angle.
- To delete an existing additional angle, select it and click the **Delete** button.
- You will have something similar to the following:



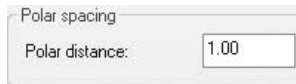
- AutoCAD will not give the multiples of the additional angles.

Polar Snap

- We previously discussed the **SNAP** command, which helped us to specify exact points on the X,Y plane using the mouse.
- The **SNAP** command can only help us along the X-axis (+ and -), and along the Y-axis (+ and -).
- If you want to snap to a point along a ray produced by **Polar Snap**, you have to change the type of the **SNAP** from **Grid Snap** to **Polar Snap**.
- On the **Status Bar**, switch on **SNAP**. Right-click on the **SNAP** button and select **Settings**. Under **Snap type**, select **Polar Snap** instead of **Grid Snap**, just like the following:

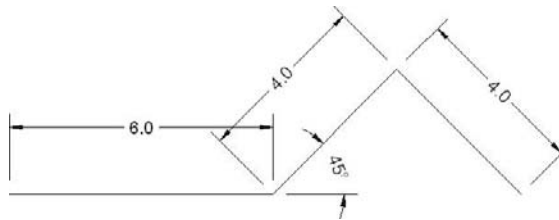


- Now, set the **Polar spacing** value, just like the following:

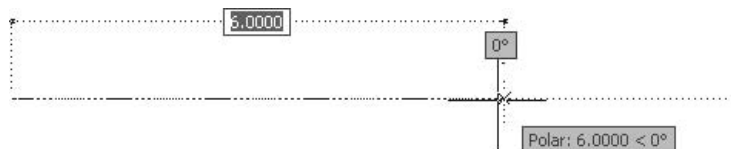


Example

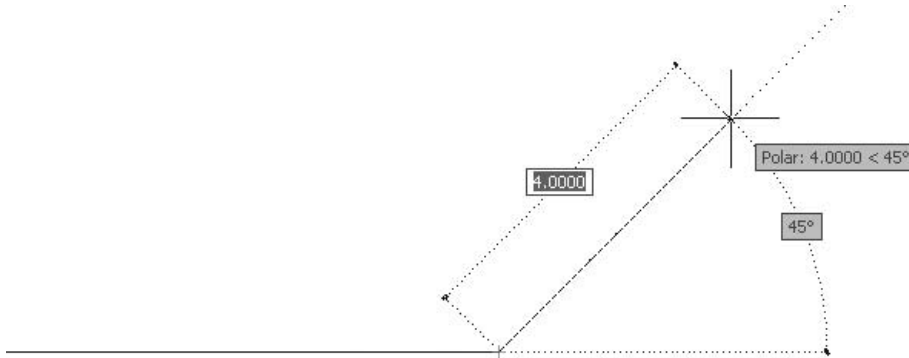
- We want to draw the following shape:



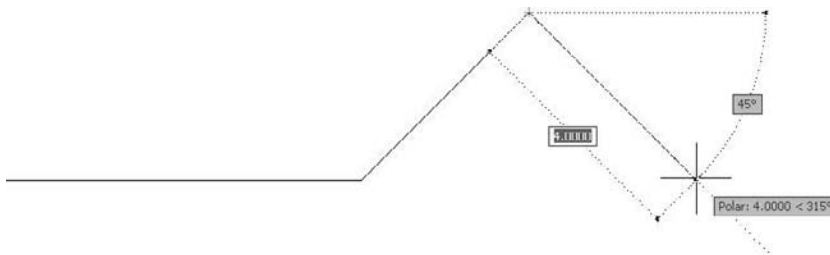
- Let's assume we set the **Increment angle** to 45 and we changed the type of **Snap** to **Polar Snap** with a distance = 1.0. The following steps show you how to draft using **Polar Tracking**:
 - Start the **Line** command, then specify a starting point.
 - Move to the right until you see the ray. Read the distance and the angle. When you reach your distance, click to specify a point just like the following:



- Move the cursor toward the angle 45 until you see the ray. Then, move the mouse to the desired distance and click:



- Move the cursor toward the angle 315 until you see the ray. Then, move the mouse to the desired distance and click:

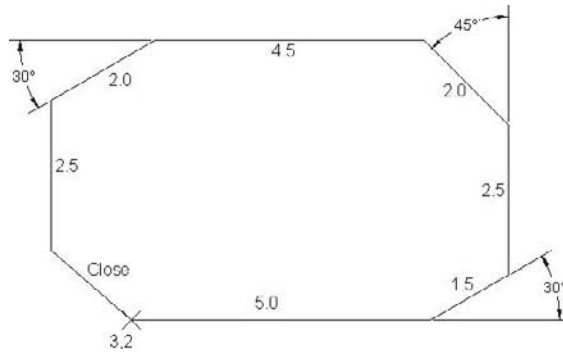


DRAWING USING POLAR TRACKING



Exercise 10

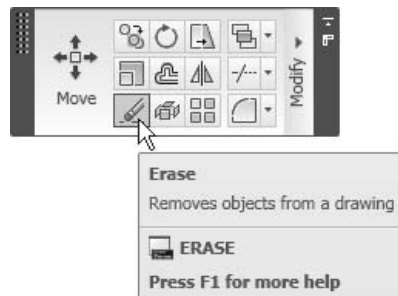
1. Start AutoCAD 2010.
2. Open the file *Exercise_10.dwg*.
3. Switch both **Polar Tracking** and **Polar Snap** on and set the following:
 - a. Increment angle = 30
 - b. Additional angles = 135
 - c. Polar distance = 0.5
4. Draw the following shape (without dimensions) starting from 3,2:



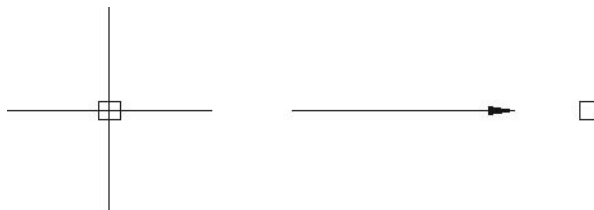
5. Save the file and close it.

2.12 THE ERASE COMMAND

- The **Erase** command will be the first modifying command we discuss.
- The only purpose of this command is to delete any object you select.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, click the **Erase** button.



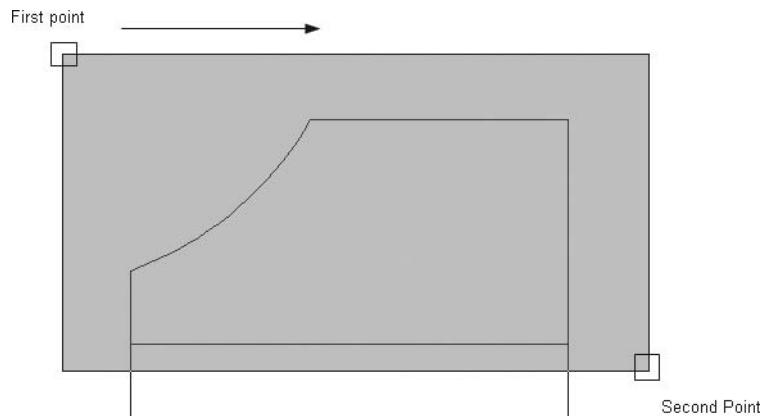
- The following prompt will appear:
Select objects:
- Once this prompt appears the cursor will change to a pick box:



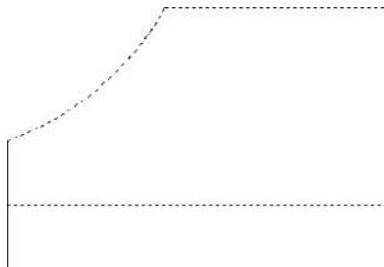
- Basically, you can do three things with the pick box:
 - Touch an object and click to select it.
 - Go to an empty place, click, and go to the right; this will get you a **Window**.
 - Go to an empty place, click, and go to the left; this will get you a **Crossing**.

Window

- A window is a rectangle specified by two opposite corners (points). The first point will be placed when you click on the empty place. Then, release your hand, go to a suitable place, and click the second point to create the opposite corner.

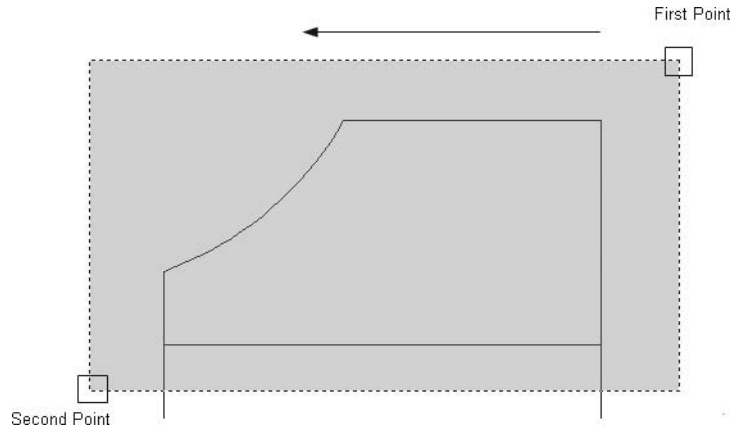


- Whatever is fully inside the rectangle will be selected. If any part (even a small part) is outside the rectangle it will not be selected. See the following illustration:

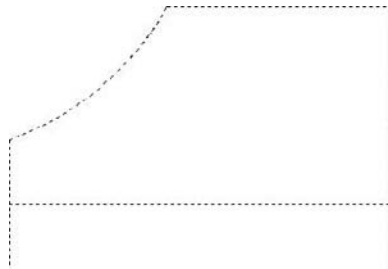


Crossing

- A crossing is just like a window, except that whatever is inside it will be selected in addition to whatever it touches.

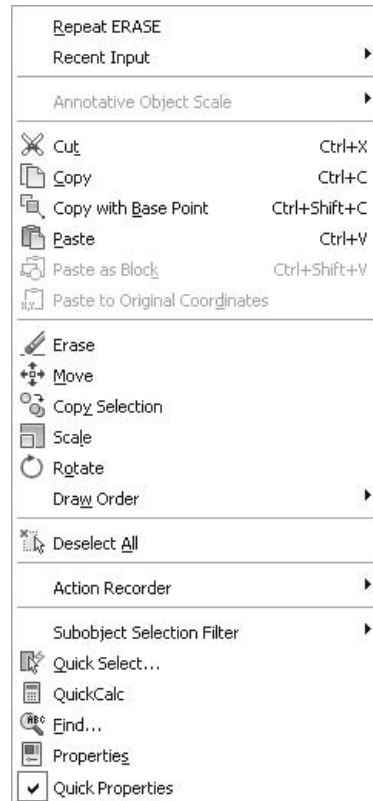


- The result will be:



- These three methods can be used with almost all of the modifying commands, not just the **Erase** command.
- The **Select Objects** prompt is repetitive. You need to always finish by pressing [Enter] or you can right-click and use the menu.
- Other ways to erase objects are:
 - Without issuing any command, click on the desired object(s) and press the [Del] key.

- Without issuing any command, click on the desired object(s), then right-click, and the following shortcut menu will appear. Select **Erase**.



2.13 OOPS, UNDO, AND REDO COMMANDS

- This group of commands can help you correct your mistakes.
- They can be used in the current session only (i.e., once you close the file, they will be useless).

Oops

- Oops** is used to retain the last group of erased objects.
- Oops** only works with the **Erase** command.
- This command is only available in the **Command Window**.
- You have to type the full command in the **Command Window**: **Oops**.
- No prompt will be displayed, but you will see the last group of erased objects returned to the drawing.

Undo

- **Undo** is used to undo the last command.
- You can reach this command using one of the following methods:
 - From the **Quick Access** toolbar, click the **Undo** button:



- Type **u** in the **Command Window** (don't type **undo**, because it has a different meaning).
- Press [Ctrl] + Z.
- The last command will be undone.
- You can undo as many commands as you want in the current session.

Redo

- This command is used to undo the undo.
- You can use this command using one of the following methods:
 - From the **Quick Access** toolbar, click the **Redo** button.



- Type **redo** in the **Command Window**.
- Or press [Ctrl] + Y.
- The last undone command will be redone.
- You can redo as many commands as you want in the current session.

2.14 REDRAW AND REGEN COMMANDS

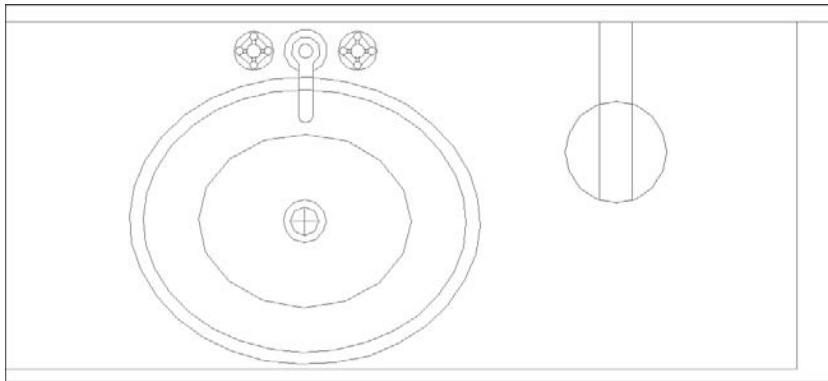
- There are times when you need to refresh the screen for one reason or another.
- Sometimes you will need AutoCAD to regenerate the whole drawing to show the arcs and circles as smooth curves.
- Neither command has a toolbar button.

Redraw

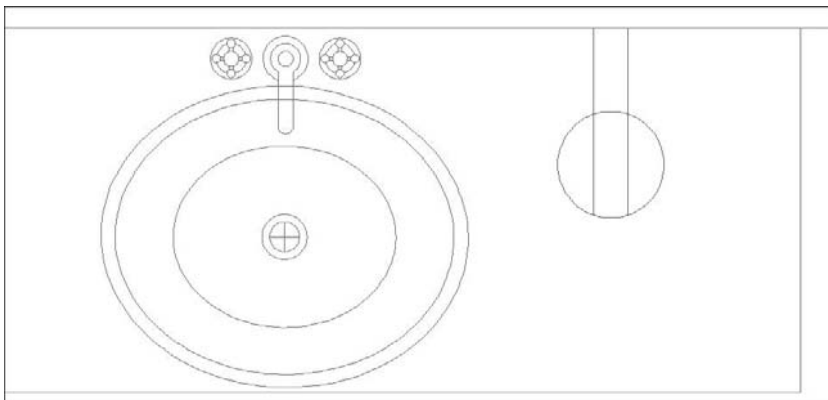
- From the **Menu Browser**, select **View/Redraw** or type **r** in the **Command Window**.
- The screen will be refreshed.

Regen

- From the **Menu Browser**, select **View/Regen** or type **re** in the **Command Window**.
- See the following example:
 - This is how the drawing looks before the **Regen** command:



- This is how the drawing looks after the **Regen** command:



- Notice the lines are smoother.

ERASE, OOPS, UNDO, AND REDO



Exercise 11

1. Start AutoCAD 2010.
2. Open the file *Exercise_11.dwg*.
3. Using the **Erase** command with **Window** or **Crossing**, perform the following steps:
 - a. Using **Window**, try to erase all of the rectangles in the middle. Press [Enter], then use the **Oops** command to retain the objects.
 - b. Using **Crossing**, try to erase the circles on the right-hand side of the shape. Press [Enter] and then **Undo**. Also, try **Redo** to see the effect.
 - c. Using a pick box, try to erase the lines of the frame. Press [Enter] and then use **Undo** to undo the erasing.
4. Close the file without saving.

CHAPTER REVIEW

1. Which of the following statements are true:
 - a. **Snap** will help us control the mouse whereas **Grid** is complementary to **Snap**.
 - b. **Ortho** and **Direct Distance Entry** will help us draw exact orthogonal lines.
 - c. You can use **Direct Distance Entry** with **Dynamic Input**, **Ortho**, and **Polar Tracking**.
 - d. All of the above.
2. The **Arc** command in AutoCAD will draw a _____ arc.
3. Using **OTRACK**, you can:
 - a. Specify a point using two existing points.
 - b. Specify the radius of an arc.
 - c. Specify the end of an existing line.
 - d. None of the above.
4. **OTRACK** doesn't need **OSNAP** to work.
 - a. True
 - b. False

5. In **Polar Tracking**, if the **Increment angle** option did not fulfill all your needs:
 - a. **Ortho** will help.
 - b. Set the additional angles.
 - c. The command **PolarNewAngles** will help.
 - d. None of the above.
6. There are _____ ways to draw a circle in AutoCAD.

CHAPTER REVIEW ANSWERS

1. d
2. circular
3. a
4. b
5. b
6. six

Chapter 3 HOW TO SET UP YOUR DRAWING

In This Chapter

- ◇ Things to Consider before You Set Up Your Drawing
- ◇ Step 1: Drawing Units
- ◇ Step 2: Drawing Limits
- ◇ Step 3: Creating Layers
- ◇ Layer Functions
- ◇ Quick Properties, Properties, and Match Properties

3.1 THINGS TO CONSIDER BEFORE YOU SET UP YOUR DRAWING

- There are many things you will need to think about when you set up your drawing file. Of course, we cannot cover them all in this chapter, but we will cover the most important things.

Drawing Units

- We will first define the drawing distance and angle units, along with their precision.

Drawing Limits

- Try to figure out what size (area) workspace will be sufficient to accommodate your drawing.

Layers

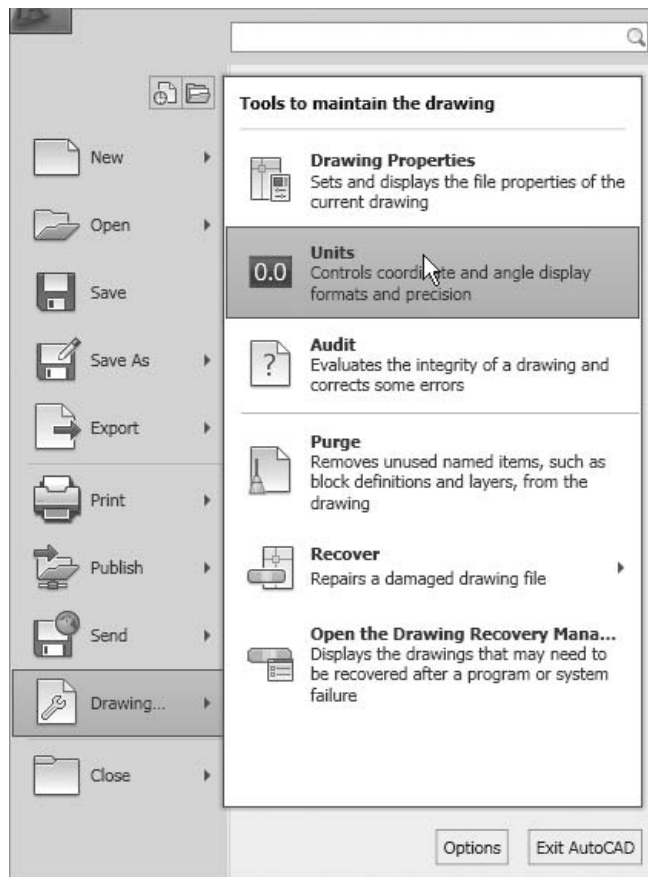
- Layers are the most effective way to organize your drawings, so we will learn what they are, how to create them, and how to control them.



- In Appendix A, we will discuss ways to create templates in AutoCAD®, which are more applicable for businesses than individuals.

3.2 STEP 1: DRAWING UNITS

- First, you will learn how to draw units.
- This command will allow you to select the proper length and angle units.
- From the **Application Menu** select **Drawing/Units**:



- After you make your selection, the following dialog box will appear:



- Under **Length**, set the desired **Type**. You will have five choices:
 - Architectural (example: 1'-5 3/16")
 - Decimal (example: 20.4708)
 - Engineering (example: 1'-4.9877")
 - Fractional (example: 17 1/16)
 - Scientific (example: 1.6531E+01)
- Under **Angle**, set the desired **Type**. You will have five choices:
 - Decimal Degrees (example: 45.5)
 - Deg/Min/Sec (example: 45d30'30")
 - Grads (example: 50.6g)
 - Radians (example: 0.8r)
 - Surveyor's Units (example: N 45d30'30" E)
- For the desired **Length** and **Angle**, select the **Precision**, for example:
 - Architectural precision can be 0'-0 1/16", 0'-0 1/32", etc.
 - Decimal precision can be 0.00, 0.000, etc.
 - Deg/Min/Sec precision can be 0d00'00", 0d00'00.0", etc.
- By default, AutoCAD deals with the positive angles counterclockwise. If you prefer it the other way around, check the **clockwise** box.

- Under **Insertion scale**, specify **Units to scale inserted content**, which is your drawing's scale against the scale of any object (a block, for example). This will help AutoCAD make the suitable conversion.
- Click the **Direction** button to see the following dialog box:



- As we discussed in Chapter 1, AutoCAD always starts the zero angle measuring from the east. If you want to change the direction, select the desired angle to be considered as the new zero.

3.3 STEP 2: DRAWING LIMITS

- In Chapter 1, we learned that AutoCAD offers an unlimited drawing sheet, which extends in all directions. We will not use it all, instead we will specify an area that gives us our **limits**.
- **Drawing Limits** is the workspace you select to work in and can be specified using two points: the lower left-hand corner and the upper right-hand corner.
- Since we will draw in **Model Space** and print from **Paper Space**, we do not need to think about drawing scale at this point.
- To know the exact limits needed, make sure you have the following information:
 - What is the longest dimension in your sketch in both the X-axis and Y-axis?
 - What AutoCAD unit have you selected (e.g., meter, centimeter, millimeter, inch, foot, etc.)?
- Accordingly, you will know the limit of your drawing.

Example

- Assume we have the following case:
 - We want to draw an architectural plan, which extends in X for 50 m and in Y for 30 m.
 - Also, assume that one AutoCAD unit = 1 m.

- If one AutoCAD unit = 1 m, then 50 m is equal to 50 AutoCAD units, which also applies to 30 m.
- Note that 0,0 is always the common lower left-hand corner, so there is no need to change it. The upper right-hand corner will be 50,30.
- At the command prompt, type **limits**, the following prompt will appear in the **Command Window**:

Specify lower left corner or [ON/OFF] <0,0>: (**press [Enter] to accept the default value**) Specify upper right corner <12,9>: (**type in the coordinate of the upper right corner**)



- To keep yourself from using any area outside this limit, turn on the **Limits**.

DRAWING UNITS AND LIMITS



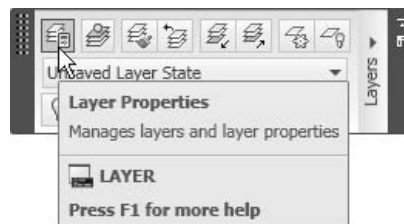
Exercise 12

1. Start AutoCAD 2010.
2. Open the file *Exercise_12.dwg*.
3. Note the current units (look at the lower left-hand corner of the screen and you will see the coordinates of the drawing).
4. From the **Application Menu** select **Drawing/Units**. Change the units to be:
 - a. Length Type = Architectural
 - b. Length Precision = 0'-0 1/32"
 - c. Angle Type = Deg/Min/Sec
 - d. Angle Precision = 0d00'00"
5. Now, check the coordinates again and see how the numbers have changed with the new units.
6. Using the **Limits** command, do the following:
 - a. Accept the default point for the lower left-hand corner.
 - b. For the upper right-hand corner type **30',20'**.
7. Switch **Grid** on and double-click on the mouse wheel.
8. You will see your current settings.
9. Save and close the file.

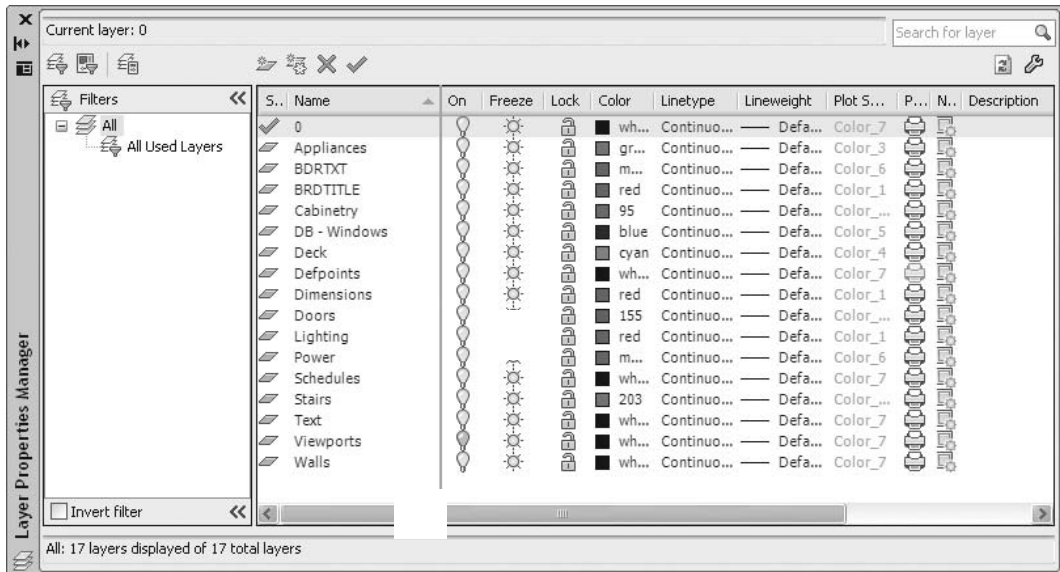
3.4 STEP 3: CREATING LAYERS

What Are Layers?

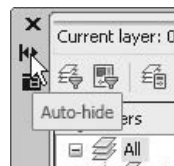
- Let's assume that we have a large number of transparent papers along with 256 colored pens.
- Taking care that we do not draw anywhere except on the top of the paper, we select the red pen and draw the border of the drawing.
- Then, we move the second paper to the top, and we draw an architectural wall plan using the yellow pen.
- Next, we move the third paper to the top and we draw the doors using the green pen. Employing the same procedure we draw windows, furniture, electrical outlets, hatching, text, dimensioning, etc.
- Then, we take all of the papers and look at them at the same time. What do we see? A full architectural plan!
- In AutoCAD we call each paper a layer.
- Each layer should have a name, color, linetype, linewidth, and much more information.
- There will be a layer, which will be in all of AutoCAD's drawings. This layer is 0 (zero). You cannot delete it or rename it.
- In order to draw on a layer, you must first make it **current**. Only one layer will be current at a time.
- The objects drawn on a layer will automatically inherit the properties (color, linetype, linewidth, etc.) of the current layer. Hence, a line in the red layer, with a dashdot linetype and 0.3 linewidth will have the exact same properties.
- By default, the setting of the object's color is = BYLAYER.
- By default, the setting of the object's linetype is = BYLAYER.
- By default, the setting of the object's linewidth is = BYLAYER.
- It is highly recommended to keep these settings intact, as changing them may lead to creating objects with nonstandard properties.
- On the **Ribbon**, make sure you are in the **Home** tab. Using the **Layers** panel, click the **Layer Properties** button.



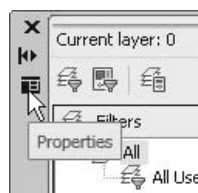
- The following dialog box will appear:



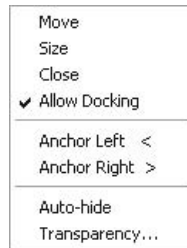
- The **Layer Properties** is not a normal dialog box; rather, it is a palette that can be docked, resized, and hidden.
 - Drag the title of the palette to the right, left, top, or bottom of the screen and you will see the **Layer Properties** palette change its size and dock at the place you select.
 - You can hide the entire palette and show only the title bar by clicking the **Auto-hide** button as shown in the following. Whenever you want to see the palette again, simply go back to the title and the palette will appear.



- You can show the **Properties** menu to control the palette. Click the **Properties** button as shown.



- The following menu will appear:



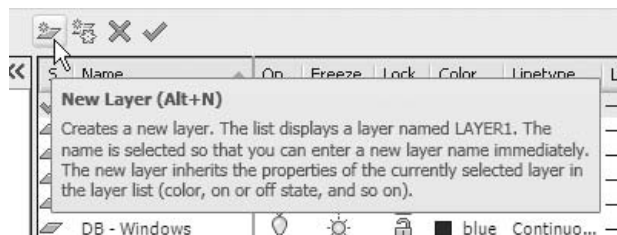
- The most important options available in this menu are **Anchor Left** and **Anchor Right**, which automatically dock the palette at the right or at the left and will switch on **Auto-hide**.
- You can resize the palette to be larger or smaller. Move to the lower right-hand corner of the palette; the cursor will change to the following:



- Click and drag to the right to make it larger. Click and drag it to the left to make it smaller.

Creating a New Layer

- To create a new layer in the drawing, you must prepare all of the necessary information for the new layer.
- Click the **New Layer** button.



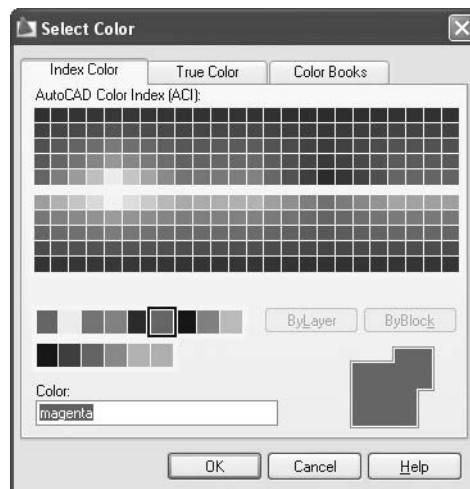
- AutoCAD will add a new layer with the temporary name *Layer1*. The **Name** field will be highlighted. Type the desired name of the layer

(you can use up to 255 characters and spaces are allowed). Only use the following:

- Letters (a, b, c, ..., z); lowercase or uppercase doesn't matter
 - Numbers (0, 1, 2, ..., 9)
 - Hyphen (-), underscore (_), and dollar sign (\$)
- It is a common practice to use good layer naming, using a name that gives an idea about the contents of the layer. For example, a layer that contains the walls of a building would be named "wall."

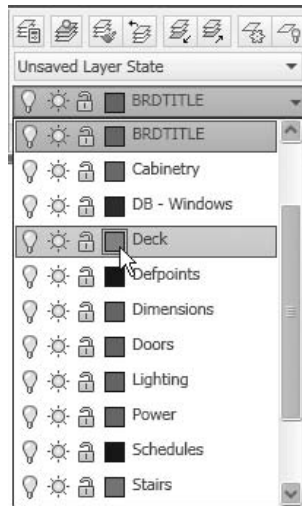
Setting a Color for a Layer

- After you create a layer, set its color.
- AutoCAD uses 256 colors for the layers (as a matter of fact, there are only 255 if we exclude the color of the **Graphical Area**).
- The first seven colors can be called by their names or numbers:
 - Red (1)
 - Yellow (2)
 - Green (3)
 - Cyan (4)
 - Blue (5)
 - Magenta (6)
 - Black/White (7)
- The remaining colors can only be called by their numbers.
- You can have the same color for more than one layer.
- Select the desired layer under the **Color** field, and click either the name of the color or the icon. The following dialog box will appear:



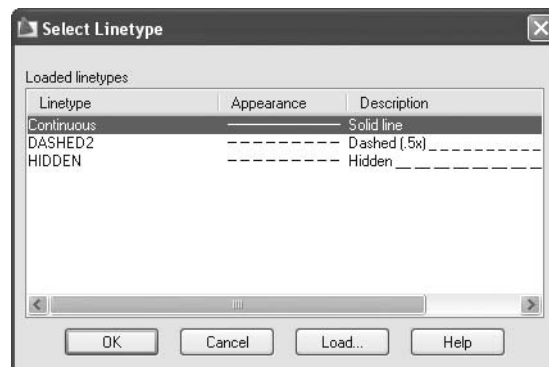


- Move to the desired color (or type in the name/number) and then click **OK**.
- You can also set the layer's color through the pop-up list in the **Layers** panel by selecting the color icon in the list:

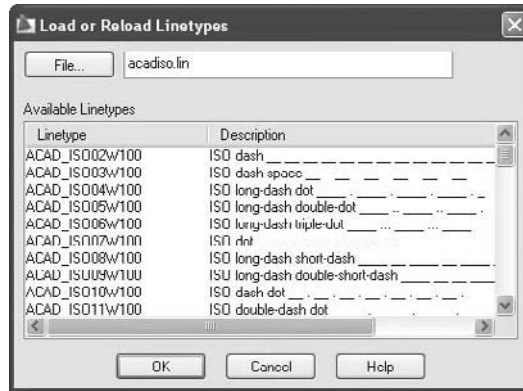


Setting a Linetype for a Layer

- AutoCAD comes with a good number of generic predefined linetypes saved in a couple of files called *acad.lin* and *acadiso.lin*.
- You can also buy other linetypes from third parties, which can be found on the Internet. Just go to any search engine and search for “AutoCAD linetype.” You will find many linetype files, some free of charge and some you can buy for few dollars.
- Not all linetypes are loaded in the drawing files; you may need to load the desired linetype first before you can use it.
- First, select the desired layer. Under the field **Linetype**, click the name of the linetype and the following dialog box will appear:



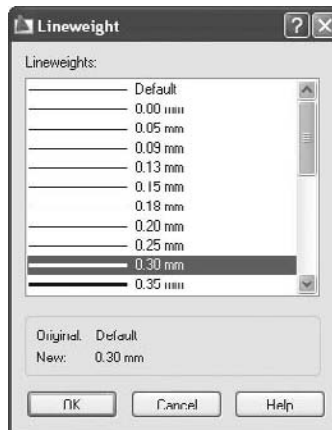
- If your desired linetype is there, select it. To load another linetype, click the **Load** button and the following dialog box will appear:



- Select the desired linetype to be loaded and click **OK**. Now that the linetype is loaded, it will appear in the **Select Linetype** dialog box. Select it and click **OK**.

Setting a Lineweight for a Layer

- Select the desired layer under the field **Lineweight**. Click either the number or the shape of the lineweight and the following dialog box will appear:



- Select the desired lineweight and click **OK**.
- If you want to view the lineweight of any layer on the screen, click the **Show/Hide Lineweight** on the **Status Bar**.

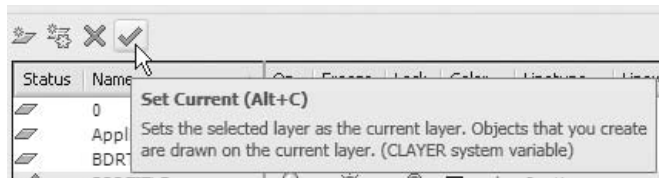




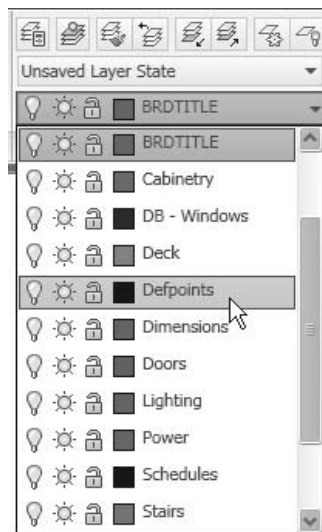
- We prefer to see the lineweight using **Plot Style** (to be discussed later), which will affect the hardcopy.

Making a Layer the Current Layer

- There are three ways to make a particular layer the current layer:
 - In the **Layer Properties Manager** dialog box, double-click on the name or the status of the desired layer.
 - In the **Layer Properties Manager** dialog box, select the desired layer and click the **Set Current** button.



- On the **Ribbon**, make sure you are in the **Home** tab, and, using the **Layers** panel, you will find a pop-up list for the layers. Select the desired layer name and it will become the current layer:



LAYER NAMES, COLORS, LINETYPES, AND LINEWEIGHTS



Exercise 13

1. Start AutoCAD 2010.
2. Open the file *Exercise_13.dwg*.
3. Create the following layers:

Layer Name	Color	Linetype	Lineweight
Shaft	Magenta	Continuous	0.3
Body	Cyan	Continuous	0.3
Base	Green	Continuous	0.3
Centerlines	9	Dashdot2	0.5

4. Make Centerlines the current layer. (Make sure **Dynamic Input** is turned off.)
5. Draw a line from 6,7.5 to 6,4.5. Draw another line from 8,6 to 4,6.
6. Save the file and close it.

3.5 LAYER FUNCTIONS

Adding More Layers

- The easiest way to add more layers is to click on the name of any layer while you are in the **Layer Properties Manager** dialog box and then press [Enter].
- Accordingly, you can use the **New Layer** button.
- By default, AutoCAD will always sort the layers according to their names.



Selecting Layers

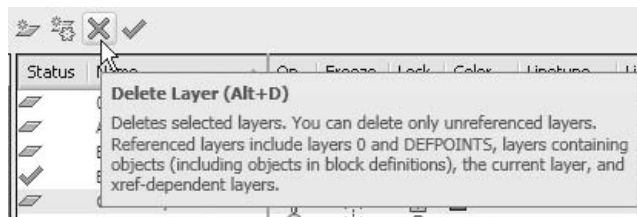
- All of the following methods will be done in the **Layer Properties Manager** dialog box.

- There are several ways to select layers:
 - To select a single layer, simply click on it.
 - To select multiple nonconsecutive layers, select the first layer, then hold the [Ctrl] key and click on the other layers.
 - To select multiple consecutive layers, select the first layer, then hold the [Shift] button and click on the last layer you wish to select.
 - To select multiple layers all at once, click on an empty area and hold the mouse. Move to the right or left and a rectangle will appear. Cover the layer that you wish to select and release the mouse.
 - To select all layers, press [Ctrl] + A.
 - To unselect a selected layer, hold the [Ctrl] key and click it.
- One of the most important advantages in selecting multiple layers is the ability to then set the color, linetype, or lineweight for group of layers in one step.

NOTE

Deleting a Layer

- You cannot delete a layer that contains objects, so the first step is to empty the layer from any objects in it.
- Using the **Layer Properties Manager** dialog box, select the desired layer (or layers) to be deleted and do one of the following:
 - Press the [Del] key on the keyboard.
 - Click on the **Delete Layer** button.



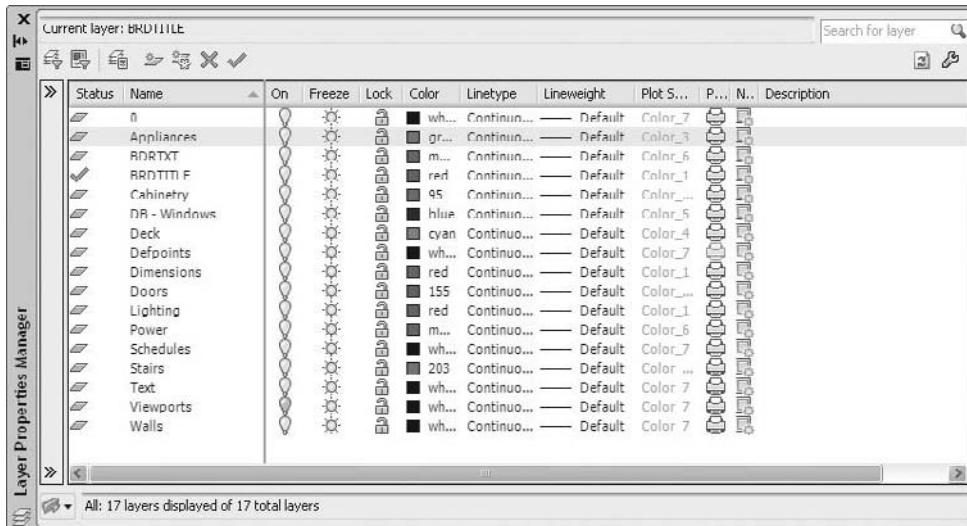
What Happens When You Right-Click?

- Right-clicking here is done in the **Layer Properties Manager** dialog box.
- If you select any layer and right-click, the following shortcut menu will appear:

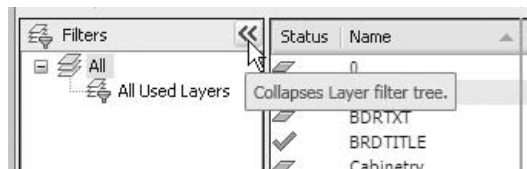


- Through this shortcut menu you can do many of the things we discussed earlier, such as:
 - Set the current layer
 - Create a new layer
 - Delete a layer
 - Select all layers
 - Clear the selection
 - Select “All but Current”
 - Invert the selection (make the selected unselected, and vice versa)
- The first two choices in this shortcut menu are:
 - **Show Filter Tree** (turned on by default)
 - **Show Filters in Layer List** (turned off by default)

- By turning off the **Show Filter Tree**, the dialog box will have more space, just like the following:



- Or you can use the two arrows at the left panel, as in the following:



Changing an Object's Layer

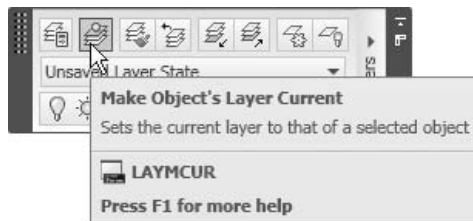
- Each object should exist in a layer.
- The fastest way to change the object's layer is the following:
 - Without issuing any command, select the object by clicking it.
 - In the **Layers** toolbar, the object's layer will be displayed. To change it, click the layer's pop-up list and select the new layer.
 - Press the [Esc] key one time.
- Other methods to change an object's layer will be discussed later.



Making an Object's Layer Current

- This function is very useful when there are too many layers in your drawing or you see an object in your drawing, but you do not know in which layer this object resides in.

- What you want to do is to make this object's layer the current layer. To do so you must:
 - Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Layers** panel, click the **Make Object's Layer Current** button.



- The following prompt will appear:

Select object whose layer will become current:

(click on the desired object)

Walls is now the current layer.

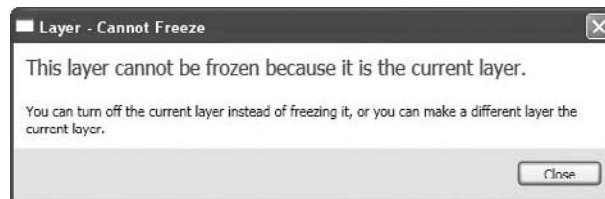
- Now the current layer is the object's layer.

What Are the Four Switches of a Layer?

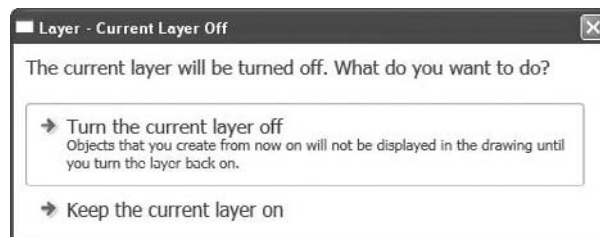
- Each layer has four switches, which determine its state.
- You can see these switches in both the **Layer Properties Manager** dialog box and the layer pop-up list from the **Layer** panel.
- These switches are:
 - On/Off
 - Thaw/Freeze
 - Unlock/Lock
 - Plot/No Plot
- See the following example:

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Plot S...	P...	N...	Descr
	0				wh...	Continuo...	Default	Color_7			
	Appliances				gr...	Continuo...	Default	Color_3			
	BDRTXT				m...	Continuo...	Default	Color_6			
	BRDTITLE				red	Continuo...	Default	Color_1			
	Cabinetry				95	Continuo...	Default	Color_...			
	DB - Windows				blue	Continuo...	Default	Color_5			
	Deck				cyan	Continuo...	Default	Color_4			
	Defpoints				wh...	Continuo...	Default	Color_7			
	Dimensions				red	Continuo...	Default	Color_1			
	Doors				155	Continuo...	Default	Color_...			
	Lighting				red	Continuo...	Default	Color_1			
	Power				m...	Continuo...	Default	Color_6			
	Schedules				wh...	Continuo...	Default	Color_7			
	Stairs				203	Continuo...	Default	Color_...			
	Text				wh...	Continuo...	Default	Color_7			
	Viewports				wh...	Continuo...	Default	Color_7			
	Walls				wh...	Continuo...	Default	Color_7			

- In the preceding example, you can see that:
 - The **Appliances** layer is On, Thaw, Unlocked, and Plot.
 - The **Cabinetry** layer is Off, Frozen, Locked, and No Plot.
- These four switches are independent from each other.
- By default, the layers are On, Thaw, Unlock, and Plot.
- When you turn a layer off, the objects in it will not be shown on the screen, and if you plot the drawing, the objects will not be plotted. However, the objects in this layer will be counted in the total count of the drawing and the drawing size will not change.
- When you freeze a layer, the objects in it will not be shown on the screen, and if you plot the drawing, the objects will not be plotted. Also, the objects in this layer will not be counted in the total count of the drawing; therefore, the drawing size will be less.
- When you lock a layer, none of the objects in it are modifiable.
- When you make a layer No Plot, you can see the objects on the screen, but when you issue a **Plot** command, these objects will not be plotted.
- Three of these switches can be changed using both the **Layer Properties Manager** dialog box, and the layer pop-up list from the **Layer** panel. The fourth switch, Plot/No Plot, can be changed only from the **Layer Properties Manager** dialog box.
- To change the switch, simply click it.
- You cannot freeze the current layer, but you can turn it off. See the following dialog box:

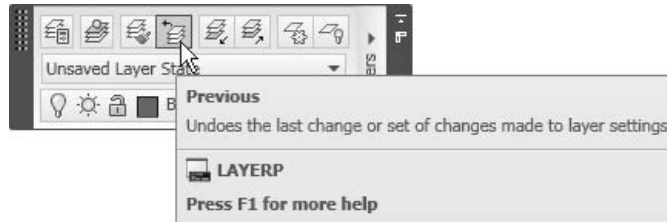


- You should be careful when you turn the current layer off because each and every time you draw a new object it will disappear. Therefore, when you try to turn the current layer off, AutoCAD will issue the following warning message:



What Is Layer Previous?

- While you are working in AutoCAD, you will change the state of layers a lot, which means you need a tool to help you return to the previous state quickly.
- **Layer Previous** helps you do that.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Layers** panel, click the **Previous** button:



- AutoCAD will report the following statement:

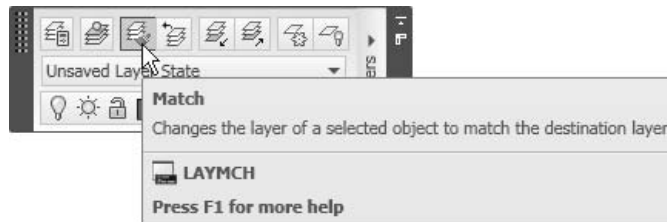
Restored previous layer status



- While you are in the **Layer Properties Manager** palette or layer pop-up list from the **Layer** toolbar, and you make several changes on several switches for several layers, AutoCAD considers them all as one action. Thus, they will all be restored in one **Previous** command.

What Is Layer Match?

- **Layer Match** converts objects from one layer to another.
- You can use the **Layer Match** tool to help you unify objects belonging to different layers.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Layers** panel, click the **Match** button.



- The following prompt will appear:

Select objects to be changed: (*Select the desired objects and once you are done, press [Enter]*)

Select object on destination layer or [Name]:

- Once the command is done, you will see a message similar to this:

8 objects changed to layer "Dimensions"

LAYER FUNCTIONS



Exercise 14

1. Start AutoCAD 2010.
2. Open the file *Exercise_14.dwg*.
3. Change the object's layers as follows:
 - a. Change the layer of the two circles from 0 to Shaft.
 - b. Change the layer of the two arcs from 0 to Body.
 - c. Change the layer of the lines from 0 to Base.
4. Using the **Status Bar**, switch on **Show/Hide Lineweight**, to see the objects displaying the assigned lineweight.
5. Lock the layer Shaft and then try to erase the objects in it. What message do you receive from AutoCAD?
6. Unlock the layer Shaft.
7. Using the **Make Layer Object's Current** button, select one of the centerlines. Which layer becomes current?
8. Click the **Layer Previous** button twice. What happens?
9. Try to freeze the current layer. What message do you receive from AutoCAD?
10. Try to rename layer 0? What message do you receive from AutoCAD?
11. Rename layer Centerlines to be Center_lines.
12. Try to delete the layer Shaft. What message do you receive from AutoCAD? Why?
13. Save the file and close it.

3.6 QUICK PROPERTIES, PROPERTIES, AND MATCH PROPERTIES

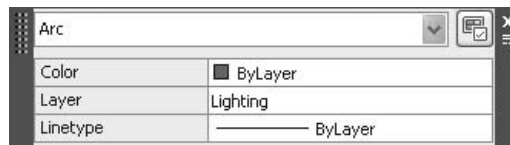
- Earlier in this chapter, we learned that each object inherits the properties of the layer that it resides in. By default, the settings of the current color, linetype, and lineweight is BYLAYER, which means that the object follows the layer it resides in.
- This makes controlling the drawing easier, because it is easier to control a handful of layers than it is to control hundreds of thousands of objects. We recommend you do not change these settings under normal circumstances.
- However, sometimes we may need to change some of the properties. To do that we can use three commands. They are:
 - **Quick Properties**
 - **Properties**
 - **Match Properties**

Quick Properties

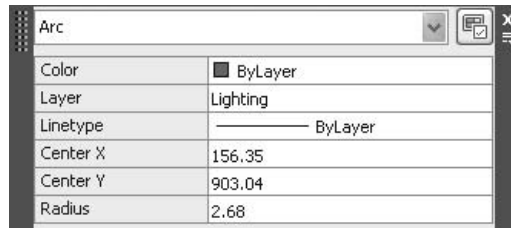
- **Quick Properties** is a function that will pop up automatically when you select any object.
- By default, this command is always on, if not, you can turn it on from the **Status Bar**:



- To start the **Quick Properties** command simply click any object and the following small panel will appear:

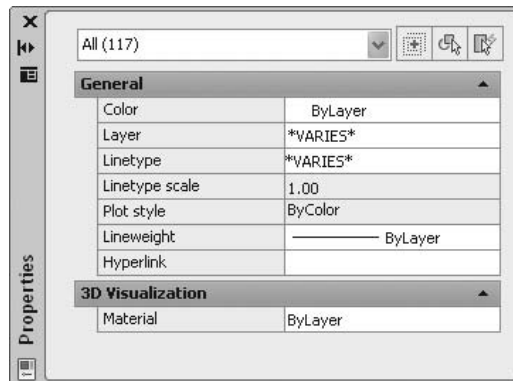


- In this panel you can change the color, layer, and linetype. If you move the mouse over any of the two sides, it will expand just like the following:

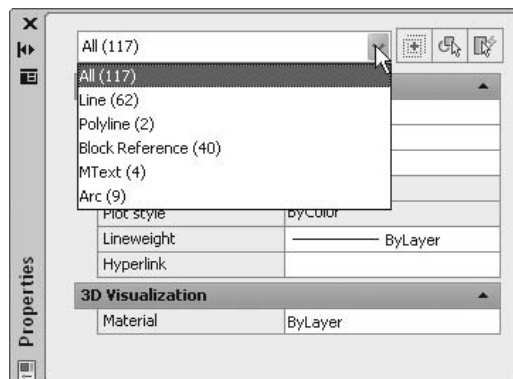


Properties

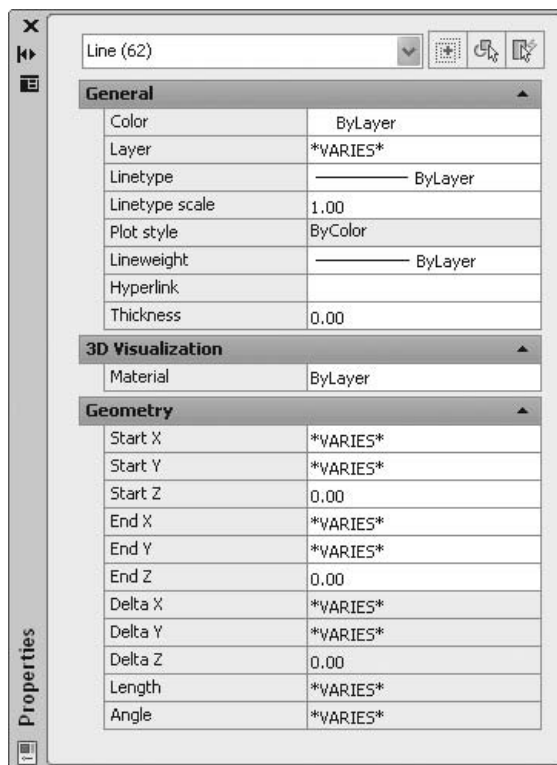
- The easiest way to initiate the **Properties** command is to select the desired object(s) and then right-click. When the shortcut menu appears, select **Properties**. Now you will be presented with two possibilities:
 - The selection set you made consists of different object types (lines, arcs, circles, etc.). In this case, you can only change the general properties of these objects. The following will appear:



- However, if you choose the upper pop-up list, you will see the following:



- You can filter the objects you select by selecting the type of object desired. You can change any or all of the properties.
- The selection set you made consists of a single object type. In this case, you can change the general properties and the object-specific properties. The following will appear:



- **Properties** is a palette, which means all the things we learned about the **Layer** palette are applicable here.

Match Properties

- **Match Properties** is useful if you opened a drawing and found that the creator of the drawing did not use the BYLAYER.
- For example, you find a green line residing in a red layer and a dashdot circle in a layer with continuous linetype.
- The best way to correct this is to try to find one object in each layer that has the right properties and then match the other objects to it.

- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Clipboard** panel, click the **Match Properties** button.



- AutoCAD will give the following prompt:

Select source object:

- Click on the object that has the right properties.
- The cursor will change to a brush shape:



- AutoCAD will then prompt:

Select destination object(s):

- Click on the objects you want to correct. Once you are done, press [Enter].

QUICK PROPERTIES, PROPERTIES, AND MATCH PROPERTIES



Exercise 15

- Start AutoCAD 2010.
- Open the file *Exercise_15.dwg*.
- We accidentally drew all objects in layer 0. Using all of the commands you have learned, put each object in its correct layer.
- Save the file and close it.

CREATING OUR PROJECT (METRIC)



Workshop 1-A

- Start AutoCAD 2010.
- Close any open files.
- Create a new file based on the *acad.dwt* template.

4. Double-click on the mouse wheel to **Zoom Extents**.
5. From the **Application Menu**, select **Drawing/Units** and make the following changes:
 - a. Length Type = Decimal, Precision = 0
 - b. Angle Type = Decimal Degrees, Precision = 0
 - c. Unit to scale inserted content = Millimeters
6. Assume that one AutoCAD unit = 1 mm and you have a 30 x 20 m plan you want to draw. With these measurements, your limits will be:
 - a. Lower left-hand corner = 0,0
 - b. Upper right-hand corner = 30000,200000
7. Type **Limits** in the **Command Window**, and set the limits accordingly.
8. Double-click on the mouse wheel to **Zoom Extents** to the new limits.
9. Create the following layers:

Layer Name	Color	Linetype	Special Remarks
Frame	Magenta	Continuous	
Walls	Red	Continuous	
Doors	Yellow	Continuous	
Door_Swing	Yellow	Dashed	
Windows	150	Continuous	
Centerlines	Green	Dashdot	
Bubbles	Green	Continuous	
Furniture	41	Continuous	
Staircase	140	Continuous	
Text	Cyan	Continuous	
Hatch	White	Continuous	
Dimension	Blue	Continuous	
Viewports	8	Continuous	No Plot

10. Save the file in the Metric folder (in the copied folder from the DVD) as *Workshop_01.dwg*.

CREATING OUR PROJECT (IMPERIAL)



Workshop 1-B

1. Start AutoCAD 2010.
2. Close any open files.
3. Create a new file based on the *acad.dwt* template.
4. Double-click on the mouse wheel to **Zoom Extents**.
5. From the **Application Menu**, select **Drawing/Units** and make the following changes:
 - a. Length Type = Architectural, Precision = 0'-0"
 - b. Angle Type = Decimal Degrees, Precision = 0
 - c. Unit to scale inserted content = Inches
6. Assume that one AutoCAD unit = 1 inch and you have a 70' x 60' plan you want to draw. With these measurements your limits will be:
 - a. Lower left-hand corner = 0,0
 - b. Upper right-hand corner = 70',60'
7. Type **Limits** in the **Command Window** and set the limits accordingly.
8. Double-click on the mouse wheel to **Zoom Extents** to the new limits.
9. Create the following layers:

Layer Name	Color	Linetype	Special Remarks
Frame	Magenta	Continuous	
Walls	Red	Continuous	
Doors	Yellow	Continuous	
Door_Swing	Yellow	Dashed	
Windows	150	Continuous	
Centerlines	Green	Dashdot	
Bubbles	Green	Continuous	
Furniture	41	Continuous	
Staircase	140	Continuous	
Text	Cyan	Continuous	
Hatch	White	Continuous	
Dimension	Blue	Continuous	
Viewports	8	Continuous	No Plot

10. Save the file in the Imperial folder (in the copied folder from the DVD) as *Workshop_01.dwg*.

CHAPTER REVIEW

1. Layer names can:
 - a. Have up to 255 characters.
 - b. Include spaces.
 - c. Have letters, numbers, hyphens, underscores, and dollar signs.
 - d. All of the above.
2. There are _____ different length units in AutoCAD.
3. What do you need to know to set up limits in a file?
 - a. The paper size you will print on.
 - b. The longest dimension of your sketch in both X and Y.
 - c. The measure of each AutoCAD unit.
 - d. B and C.
4. Only the first seven colors can be called by name and number.
 - a. True
 - b. False
5. What is true about linetypes in AutoCAD?
 - a. They are stored in *acad.lin* and *acadiso.lin*.
 - b. They are loaded in all AutoCAD drawings.
 - c. If I need to use a linetype I have to load it first.
 - d. A and C.
6. If you assign a lineweight to a layer, and on this layer you draw lines, you need to click on _____ from the **Status Bar** to see this lineweight on the monitor.
7. You can only change the _____ properties of nonsimilar objects using the **Properties** command.

CHAPTER REVIEW ANSWERS

1. d
2. five
3. d
4. a
5. d
6. **Show/Hide Lineweight** button
7. general

A FEW GOOD CONSTRUCTION COMMANDS

In This Chapter

- ◇ Introduction
- ◇ The **Offset** Command
- ◇ The **Fillet** Command
- ◇ The **Chamfer** Command
- ◇ The **Trim** Command
- ◇ The **Extend** Command
- ◇ The **Lengthen** Command
- ◇ The **Join** Command

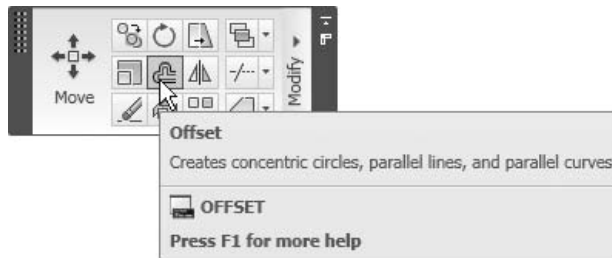
4.1 INTRODUCTION

- So far, we have learned four drawing commands (**Line**, **Arc**, **Circle**, and **Pline**).
- These alone can only help you accomplish 20% of your drawing.
- Also, if you think that each and every line (or arc, or circle) should be drawn by you, you are wrong!
- In this chapter, we will discuss seven commands that will help us construct the most difficult drawings in no time.
- These commands are:
 - The **Offset** command creates parallel copies of your original objects.
 - The **Fillet** command allows you to close unclosed shapes either by extending the two ends to an intersecting point or by using an arc.
 - The **Chamfer** command is exactly the same as the **Fillet** command, except this command will create a slanted edge.
 - The **Trim** command allows some objects to act as cutting edges for other objects to be trimmed.
 - The **Extend** command allows you to extend objects to a boundary.

- The **Lengthen** command allows you to extend or trim length from an existing line.
- The **Join** command allows you to join similar objects (lines to lines, polylines to polylines, etc.).

4.2 THE OFFSET COMMAND

- The **Offset** command will create a new object parallel to a selected object.
- The new object (by default) will have the same properties as the original object and will reside in the same layer.
- There are two methods used in offset:
 - Offset distance
 - Through point
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, click the **Offset** button.



- The following prompt will appear:

```
Current settings: Erase source=No Layer=Source
OFFSETGAPTYPE=0
Specify offset distance or [Through/Erase/Layer]
<Through>:
```

Offset Distance

- If you want to use this method, you should know the distance between the original object and the parallel duplicate (i.e., the offset distance).
- Then, select the object that will be offset.
- Finally, specify the side of the offset by clicking to the right or left, up or down, etc.
- The prompts will be as follows:

```
Specify offset distance or [Through/Erase/Layer]
<Through>: (type in the desired distance)
```

Select object to offset or [Exit/Undo] <Exit>:
(select a single object)

Specify point on side to offset or
 [Exit/Multiple/Undo] <Exit>: *(click in the desired side)*

- The command will repeat the last two prompts for further offsetting.
- To end the command, press [Enter] or right-click.

Through Point

- With this method, there is no need to know the distance but you should know any point that the new parallel object will pass through.
- The prompt will be as follows:

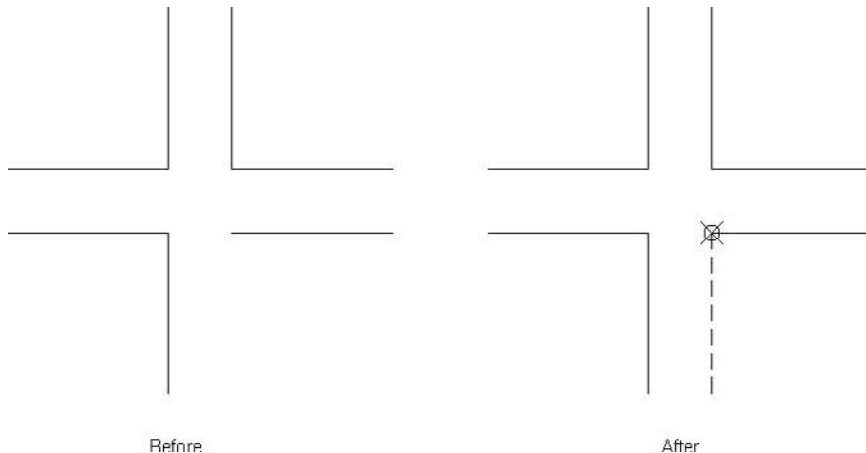
Specify offset distance or [Through/Erase/Layer]

<Through>: *(type t and press [Enter])*

Select object to offset or [Exit/Undo] <Exit>:
(Select a single object)

Specify through point or [Exit/Multiple/Undo] <Exit>:
(Specify the point that the new image will pass through)

- The command will repeat the last two prompts for further offsetting.
- To end the command, press [Enter] or right-click.
- Here is an example:



Multiple

- With both of the preceding methods you can use the **Multiple** option.
- Instead of repeatedly selecting the object and specifying the side, the **Multiple** option will allow you to specify only the side of the offset.

- The prompt will be as follows:

```
Specify offset distance or [Through/Erase/Layer]
<Through>: (Select either method)
Select object to offset or [Exit/Undo] <Exit>:
(Select a single object)
Specify through point or [Exit/Multiple/Undo] <Exit>:
(Type m and press [Enter])
Specify point on side to offset or [Exit/Undo] <next
object>: (simply click on the desired side, and you can keep
doing the same, once you are done, press [Enter])
```

Undo

- You can use the **Undo** option while you are offsetting to undo the last offsetting action.
- AutoCAD® will recall the last offset distance, so there is no need to keep typing it unless you want to use another value.
- The **Offset** command will produce a bigger or smaller arc, circle, or polyline.
- You can right-click to show shortcut menus displaying the different options of the **Command Window**.
- In the **Offset** command, you can use only one offset distance. If you want another offset distance you must end the current command and issue a new **Offset** command. (We hope to see in the next versions of AutoCAD, **Offset** commands that will allow the user to use more than one offset distance per command.)



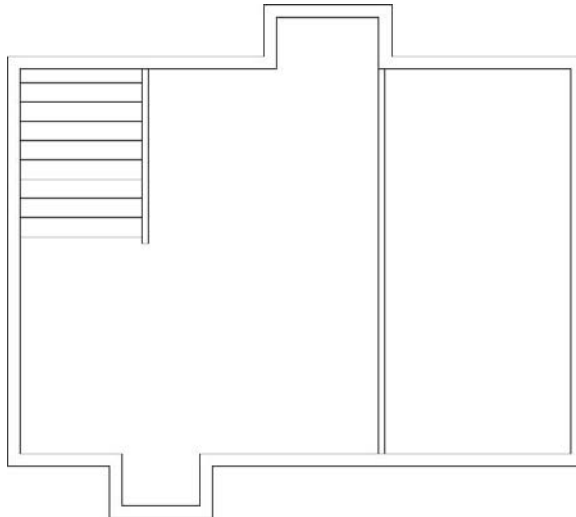
OFFSETTING OBJECTS



Exercise 16

1. Start AutoCAD 2010.
2. Open the file *Exercise_16.dwg*.
3. Offset the walls (magenta) to the inside using the distance = 1'.
4. Offset the stairs using distance = 1'-6" and using the **Multiple** option to create eight lines representing eight steps.
5. Explode the inner polyline.
6. Offset the right vertical line to the left using the through point method and the left endpoint of the upper-right horizontal line.
7. Offset the new line to the right using distance = 6".

8. The new shape of the plan should look like the following:



9. Save the file and close it.

4.3 THE FILLET COMMAND

- If you have the following:



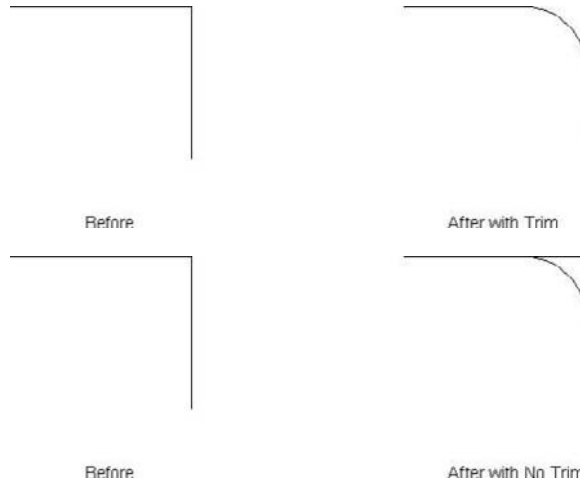
- And you want them to look like this:



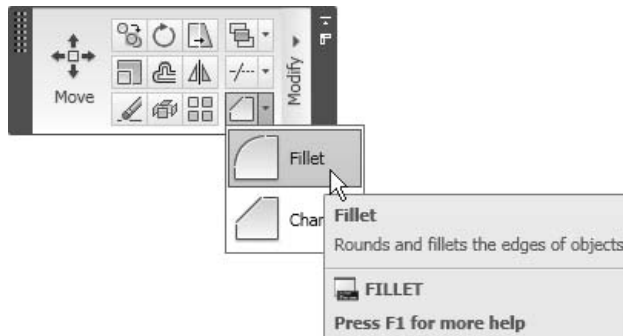
- Or, you want them to look like this:



- Then you need to use the **Fillet** command.
- Issue the **Fillet** command by selecting the first object and then the second object. It is a very simple AutoCAD command.
- The **Fillet** command works with two different settings:
 - Radius = 0 will create a neat intersection.
 - Radius > 0 will do the same except it will use an arc rather than a corner point.
- When you close the shape with an arc, what will happen to the original objects? To solve this issue, the **Fillet** command works in two different modes:
 - In **Trim** mode, the arc will be produced, and the original objects will be trimmed accordingly.
 - In **No trim** mode, the arc will be produced, but the original objects will stay intact.
- Here is an example:



- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, click the **Fillet** button.



- The following prompt will appear:

```
Current settings: Mode = TRIM, Radius = 0.0000  
Select first object or [Undo/Polyline/Radius/Trim/  
Multiple]:
```

- The first line reports the current value of the mode and the radius.
- Choose between the different options: **Radius**, **Trim**, **Multiple**, and **Undo**.

Radius

- To set a new value for the radius, the following prompt will appear:

```
Specify fillet radius <0.0000>: (type in the new radius)
```

Trim

- To change the mode from **Trim** to **No trim**, or vice versa, the following prompt will appear:

```
Enter Trim mode option [Trim/No trim] <Trim>: (type t, or n)
```

Multiple

- By default, you can perform a single fillet per command by selecting the first object and the second object.
- If you want to perform multiple fillets in a single command, you have to select the **Multiple** mode first.

Undo

- You can use the **Undo** option while you are filleting to undo the last filleting action.
- When you fillet with a radius, the radius will be created in the current layer. Make sure that you are in the right layer.
- To end the command when you use the **Multiple** option, press [Enter] or right-click.
- Even if $R > 0$, you can still fillet with $R = 0$. To do so, simply hold the [Shift] key and click on the desired objects. Regardless of the current value of R , you will fillet with $R = 0$.
- You can use the **Fillet** command to fillet two parallel lines with an arc. AutoCAD will calculate the distance between the two lines, and take the radius to be one-half of this length.

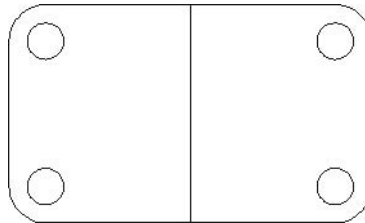


FILLETING OBJECTS

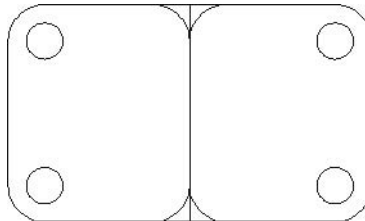


Exercise 17

1. Start AutoCAD 2010.
2. Open the file *Exercise_17.dwg*.
3. Using the **Fillet** command, perform the following steps:
 - a. Set the radius = 0.5.
 - b. Set the mode = **Trim**.
4. Fillet the four edges to make the shape look like the following:



5. Using the **Fillet** command, perform the following steps:
 - a. Mode = **No Trim**.
 - b. Set **Fillet** to be **Multiple**.
6. Fillet the lines to get the following shape:



7. Save the file and close it.

4.4 THE CHAMFER COMMAND

- The **Chamfer** command is, in many ways, identical to the **Fillet** command, except that it creates a slanted edge rather than an arc.
- To create the slanted edge, we will use one of two methods:
 - Two distances
 - Length and angle

Two Distances

- There are three different examples of this method:
 - $(\text{Dist1} = \text{Dist2}) = 0.0$, as in the following example:



- $(\text{Dist1} = \text{Dist2}) > 0.0$, as in the following example:

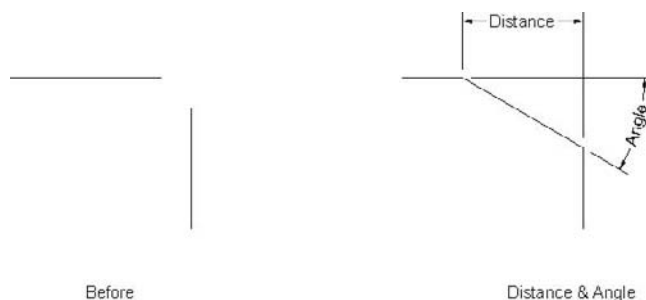


- $(\text{Dist1} \neq \text{Dist2}) > 0.0$, as in the following example (regardless of which one will be selected first, Dist1 will be used):

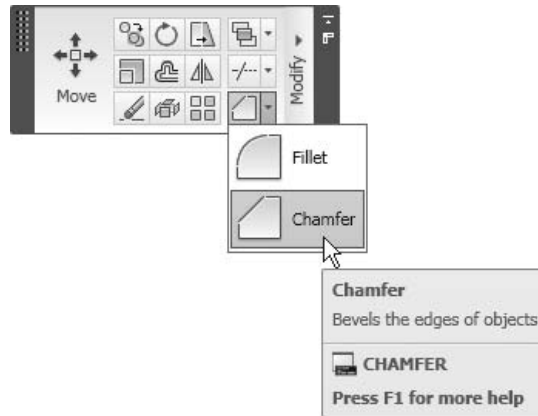


Length and Angle

- To use this method, specify a length (which will be removed from the first object) and an angle, just as in the following example:



- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, click the **Chamfer** button.



- The following prompt will appear:

```
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Undo/Polyline/Distance/Angle /Trim/
Method/Multiple]:
```

- The first line reports the current mode and the distances (or length and angle).
- Choose between the different options: distances, angle, **Trim**, method, and **Multiple**.

Distances

- To set a new value for the distances, the following prompt will appear:

```
Specify first chamfer distance <0.0000>: (input the first distance)
Specify second chamfer distance <0.0000>: (input the second distance)
```

Angle

- To set the new values for both the length and the angle, the following prompt will appear:

```
Specify chamfer length on the first line <0.0000>:
(input the length on first line)
Specify chamfer angle from the first line <0>:
(input the angle)
```

Trim

- To change the mode from **Trim** to **No trim**, or vice versa, the following prompt will appear:

Enter Trim mode option [Trim/No trim] <Trim>:
(*type t, or n*)

Method

- To specify the default method to be used in the **Chamfer** command, the following prompt will appear:

Enter trim method [Distance/Angle]
<Distance>: (*type d, or a*)

Multiple

- By default, you can perform a single chamfer per command by selecting the first object and the second object. If you want to perform multiple chamfers in a single command, you have to select the **Multiple** mode first.
- When you chamfer, the slanted line will be created in the current layer. Make sure that you are in the right layer.
- To end the command when using the **Multiple** option, press [Enter] or right-click.
- The **Trim** or **No Trim** modes in the **Fillet** command will affect the **Chamfer** command and vice versa. If you change the mode in one of these two commands, the other command will reflect this change.



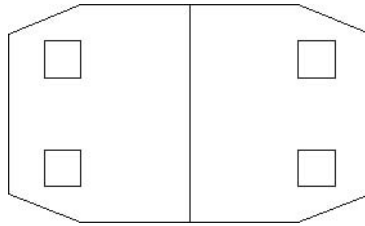
CHAMFERING OBJECTS



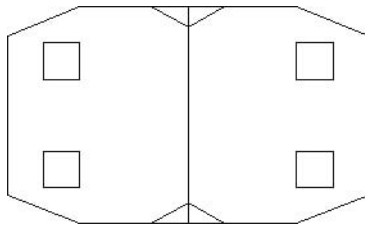
Exercise 18

1. Start AutoCAD 2010.
2. Open the file *Exercise_18.dwg*.
3. Using the **Chamfer** command, perform the following steps:
 - a. Set Dist1 = 1.0.
 - b. Set Dist2 = 0.4.
 - c. Set the mode = **Trim**.
 - d. Set the **Chamfer** command to **Multiple**.

4. Chamfer the four edges by selecting the proper line for the proper chamfering distance to make the shape look like the following:



5. Using the **Chamfer** command, perform the following steps:
 - a. Set the distance = 0.5.
 - b. Set the angle = 30.
 - c. Set the mode = **No trim**.
 - d. Set the **Chamfer** command to **Multiple**.
6. Chamfer the inner line to make the shape look like the following:

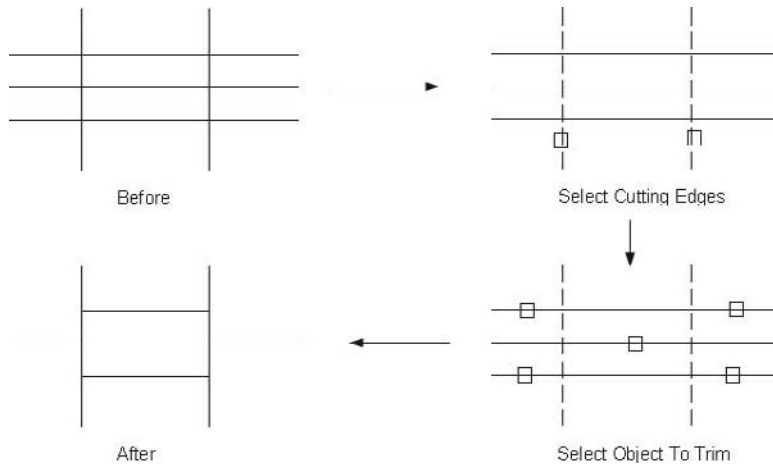


7. Save the file and close it.
 - The distance will be cut from the horizontal line and the angle will be cut from the vertical line.

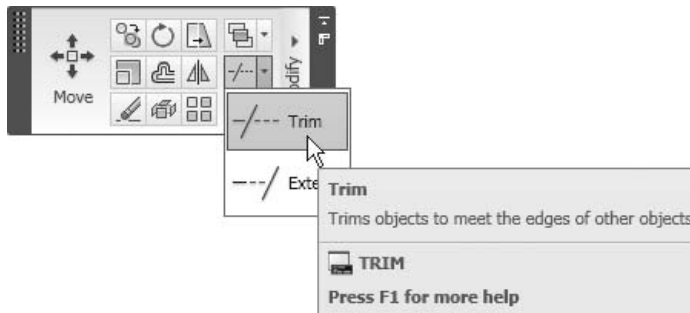
4.5 THE TRIM COMMAND

- Trimming means we want to remove part of an object by cutting the edge(s).
- The **Trim** command is a two-step command:
 - The first step is to select the edge(s) to be cut. It can be one edge or as many as you wish.
 - The second step is to select the objects to be trimmed.

- The following example illustrates the trimming process:



- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, click the **Trim** button.



- The following prompt will appear:

```
Current settings: Projection=UCS, Edge=Extend
Select cutting edges ...
Select objects or <select all>:
```

- The first line displays the current settings.
- The second line tells you to select the cutting edges.
- Use any of the methods we learned in the **Erase** command. Once you are done, press [Enter] or right-click.
- You can also use the fastest method—the **select all** option—which will select all of the objects to act as cutting edges.
- The following prompt will appear:

```
Select object to trim or shift-select to extend or
[Fence/Crossing/Project/Edge/Erase/Undo] :
```

- Now, click on the parts you want to trim, one by one.
- If you made any mistakes, simply right-click to bring up the shortcut menu and select **Undo**, or type **u** in the **Command Window**.

Fence

- You can use the **Fence** option to speed up the selection process of objects to be trimmed. This can be done by specifying two or more points. A dotted line will be created and whatever objects it touches will be trimmed.

Crossing

- The same thing applies to the **Crossing** option. When you specify two opposite corners, a crossing window will appear and any object touched by the crossing will be trimmed.

Erase

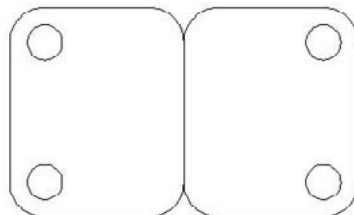
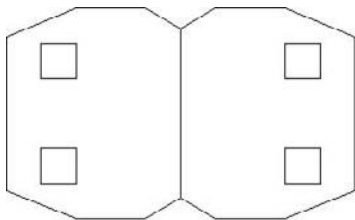
- Sometimes, as a result of trimming, there will be some unwanted objects created. Instead of finishing the command and issuing an **Erase** command, AutoCAD makes this option available for you to erase objects while you are still in the **Trim** command.
- Type **r** in the **Command Window** and AutoCAD will ask you to select the objects you want to erase. Once you are done, press [Enter], and you will be prompted again to select another option.

TRIMMING OBJECTS



Exercise 19

1. Start AutoCAD 2010.
2. Open the file *Exercise_19.dwg*.
3. Using the **Trim** command, create the following two shapes:



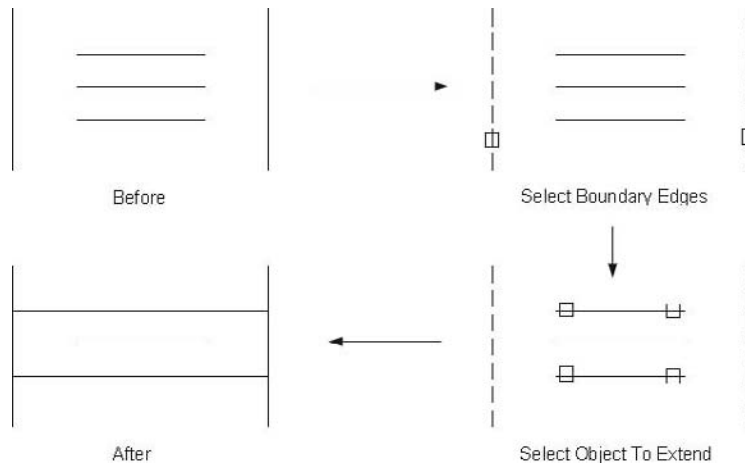
4. Save the file and close it.



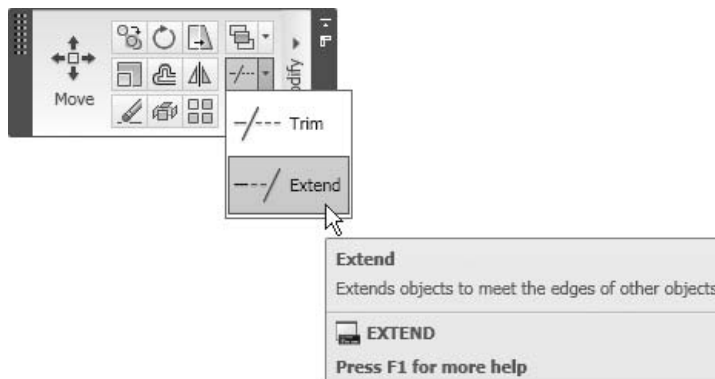
- If you get a residual object you can use the **Erase** option in the **Trim** command to get rid of it.

4.6 THE EXTEND COMMAND

- The **Extend** command is the opposite of **Trim** command.
- When you use the **Extend** command, you will extend selected objects to the boundary edge(s).
- The **Extend** command is a two-step command:
 - The first step is to select the boundary edge(s). You can choose one edge or as many as you wish.
 - The second step is to select the objects to be extended.
- The following example illustrates the process of extending:



- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, click the **Extend** button.



- The following prompt will appear:

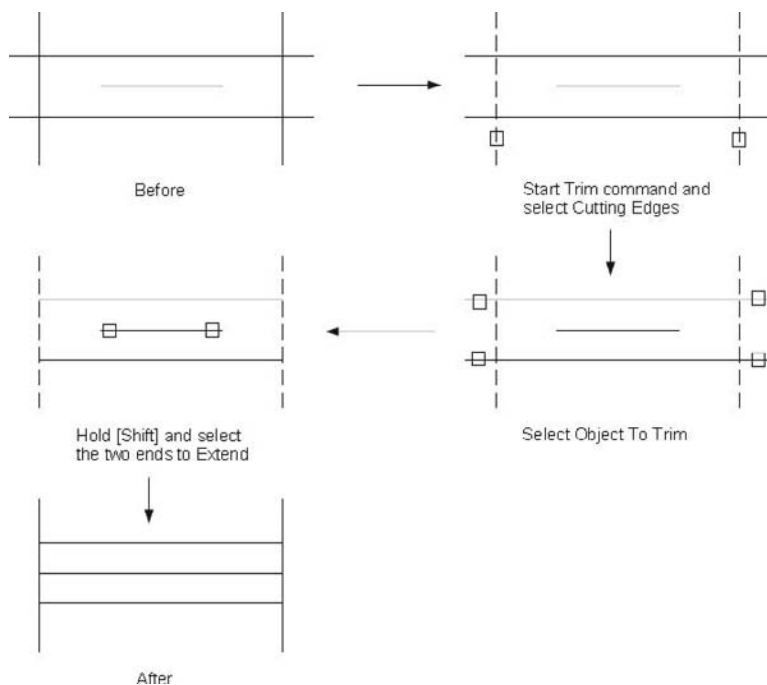
Current settings: Projection=UCS, Edge=Extend
 Select boundary edges ...
 Select objects or <select all>:

- The first line displays the current settings.
- The second line tells you to select the boundary edges.
- Use any of the methods you know. Once you are done, press [Enter] or right-click.
- The following prompt will appear:

Select object to extend or shift-select to trim or
 [Fence/Crossing/Project/Edge/Undo] :

- Now, click on the parts you want to extend, one by one.
- If you made a mistake, simply right-click to open the shortcut menu and select **Undo**, or type **u** in the **Command Window**.
- The rest of the options are the same as the **Trim** command.
- While you are in the **Trim** command, and while you are clicking on the objects to be trimmed, if you hold the [Shift] key and click, you will extend the objects rather than trim them. See the following example:

NOTE

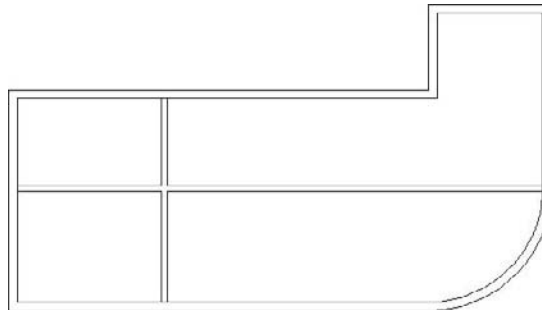


EXTENDING OBJECTS



Exercise 20

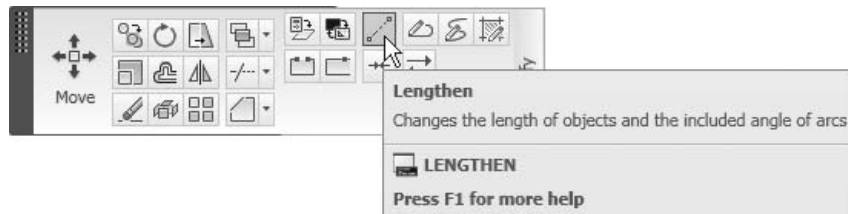
1. Start AutoCAD 2010.
2. Open the file *Exercise_20.dwg*.
3. Using the **Extend** and **Trim** commands, create the following shape:



4. Save the file and close it.

4.7 THE LENGTHEN COMMAND

- With the **Extend** command, we needed an object to serve as a boundary in order to extend the rest of the objects to it.
- The **Lengthen** (or shorten, as it serves both purposes) command will do this without a boundary.
- Make sure you are in the **Home** tab on the **Ribbon**, and, after extending the **Modify** panel, select the **Lengthen** button.



- The following prompt will appear:

Select an object or [Delta/Percent/Total/Dynamic] :

- If you click on any object, it will give you the current length.
- The command will do the lengthening (or shortening) using **Delta**, **Percent**, **Total**, or **Dynamic**.

Delta

- This command is used if you want to add or remove extra length to or from the current length.
- If you input a negative value, the **Lengthen** command will shorten the line.
- The following prompt will appear:

Enter delta length or [Angle] <0.0000>: (*input the extra length to be added*)

Percent

- If you want to add to or remove from the length, specify a percentage of the current length.
- The number should be positive, and a nonzero number. If it was > 100, it will lengthen. If it was < 100, it will shorten.
- The following prompt will appear:

Enter percentage length <100.0000>: (*input the new percentage*)

Total

- Use the **Total** option if you want the new total length of the line to be equal to the number you will input.
- If the new number is greater than the current length, the line will lengthen. If the new number is less than the current length, the line will shorten.
- The following prompt will appear:

Specify total length or [Angle] <1.0000>: (*input the new total length*)

Dynamic

- This option is used to specify a new length of the object, using the dynamic move of the mouse.
- The following prompt will appear:

Select an object to change or [Undo]: (*select the desired object*)

Specify new endpoint: (*move the mouse, up until you reach to the desired length*)



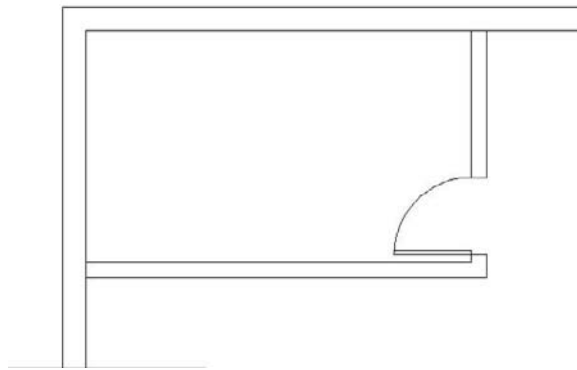
- You can use only one method per command.

LENGTHENING OBJECTS



Exercise 21

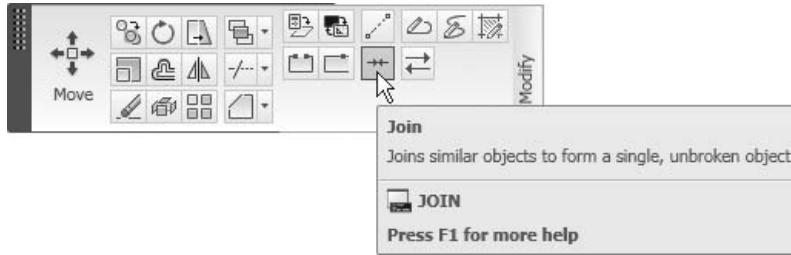
1. Start AutoCAD 2010.
2. Open the file *Exercise_21.dwg*.
3. Using the **Lengthen** command and the **Delta** option, shorten the two vertical lines by 1 unit.
4. Using the **Lengthen** command and the **Total** option, make the total length of the two horizontal lines = 5.
5. As you can see, the lower line did not come to the end like the upper line.
6. Using the **Lengthen** command and the **Percent** option, set percent = 104, and select the end of the line.
7. The output should look like the following:



8. Save the file and close it.

4.8 THE JOIN COMMAND

- While you are modifying your drawing you may end up with a line broken into segments, which you will need to join into one single line. The same thing may happen with arcs.
- If you have a polyline, you can join other objects to it, such as lines and arcs.
- Make sure you are in the **Home** tab on the **Ribbon**, and, by extending the **Modify** panel, select the **Lengthen** button.



- AutoCAD will show the following prompts:

Select source object:

Select lines to join to source: (**Select the source object**) Select lines to join to source: (**Depending on the source object, the following prompt will ask to select the objects to join, once you are done press [Enter]**) 1 line joined to source



- If you select a line object, the other lines should be collinear; gaps are allowed.
- If you select an arc object, the other arcs should be part of the same imaginary circle; gaps are allowed.
- There is a special prompt in the **Arc** option to close the arc and formulate a circle.
- You can join a polyline to any other objects (lines, arcs, etc.) but, in this case, no gaps are allowed.

JOINING OBJECTS



Exercise 22

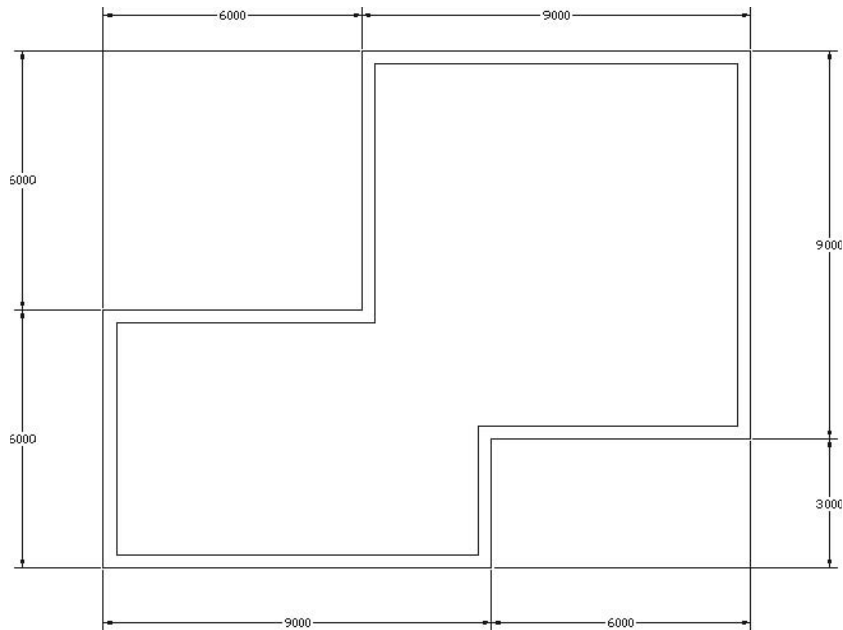
1. Start AutoCAD 2010.
2. Open the file *Exercise_22.dwg*.
3. This file displays what you might find after recovering a corrupt file.
4. You will find broken lines, arcs, and polylines.
5. Using the **Join** command, make the arc a full circle.
6. Using the **Join** command, connect the two broken lines.
7. Join all of the lines to the polyline.
8. Save the file and close it.

DRAWING THE PLAN (METRIC)

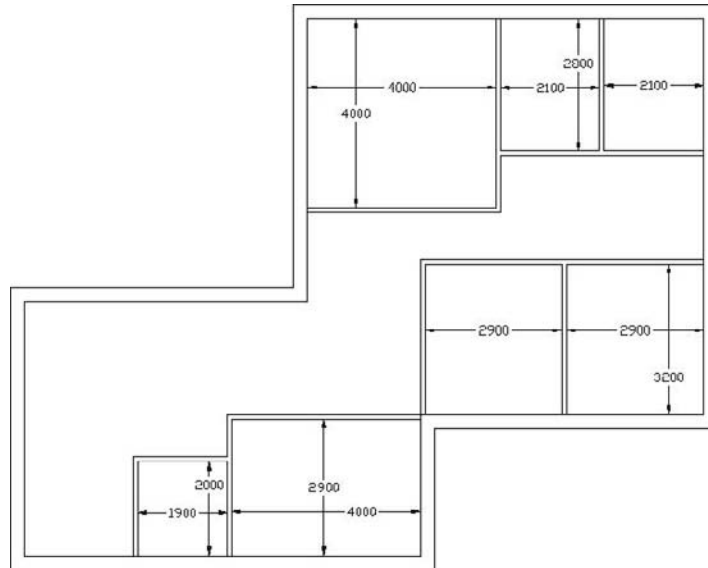


Workshop 2-A

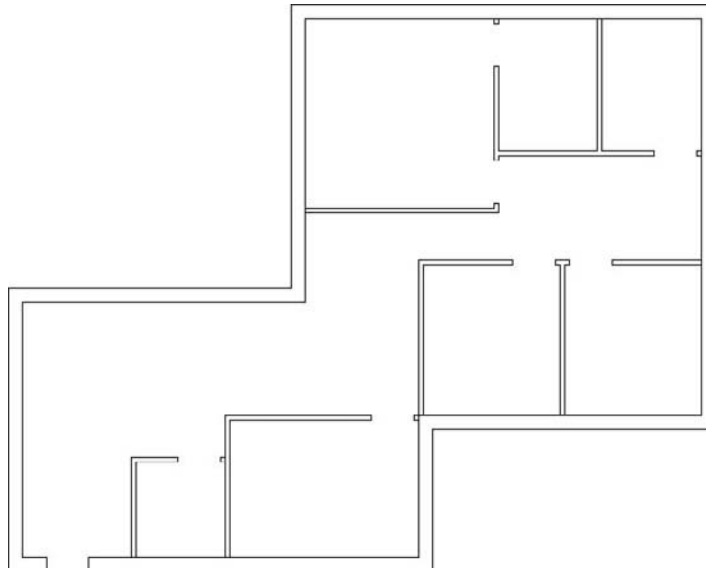
1. Start AutoCAD 2010 and open the file *Workshop_02.dwg*.
2. Make the layer **Walls** current.
3. Using the **Pline** command, draw the outer lines first (without the dimension) starting from point 8000,3000 using all the methods you learned in Chapter 2.



4. Using the **Offset** command, offset the polyline to the inside with an offset distance = 300.
5. Explode the inner polyline.
6. Using the **Offset**, **Fillet**, **Chamfer**, **Trim**, **Extend**, **Lengthen**, and **Zoom** commands, create the interior walls using the following dimensions:

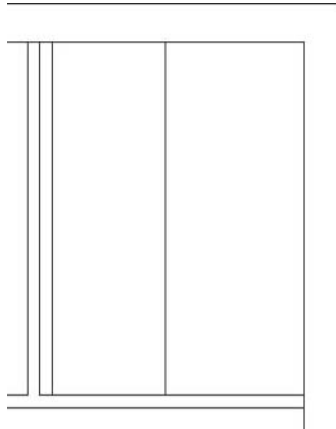


- The thickness of all inner walls = 100.
7. Make the door openings, taking the following into consideration:
 - a. All door openings = 900.
 - b. Always take 100 clear distances from the walls for the door openings (except for the outside door take 500).

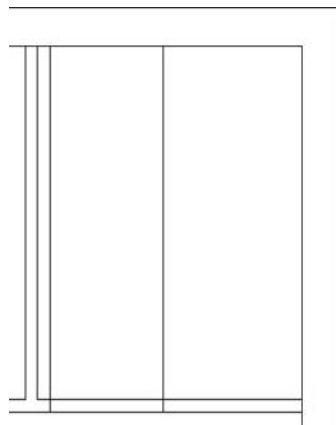




- To make the door openings, use the following technique:
 - Offset an existing wall (say, 100 for internal doors).
 - Offset the new line (say, 900 for room doors).
 - You will have the following shape:

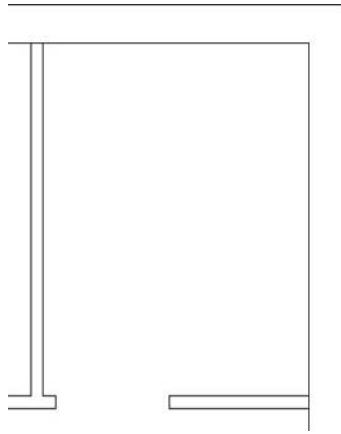


- Extend the two vertical lines to the lower horizontal line as shown in the following:



- Using the **Trim** command, select all horizontal and vertical lines as cutting edges, then press [Enter]. Trim the unnecessary parts by clicking on the objects (you can use **Crossing**, which is faster).

- This is what you will get:



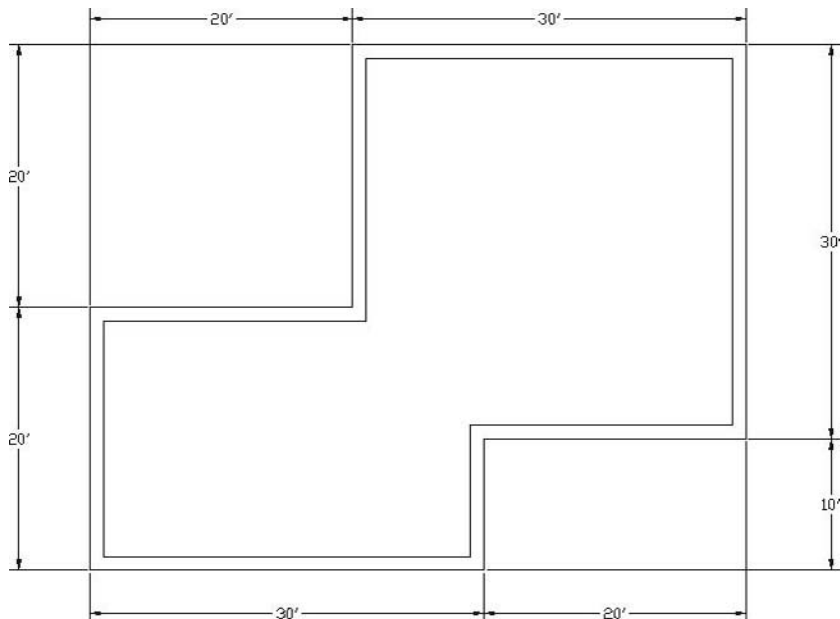
8. Save the file and close it.

DRAWING THE PLAN (IMPERIAL)

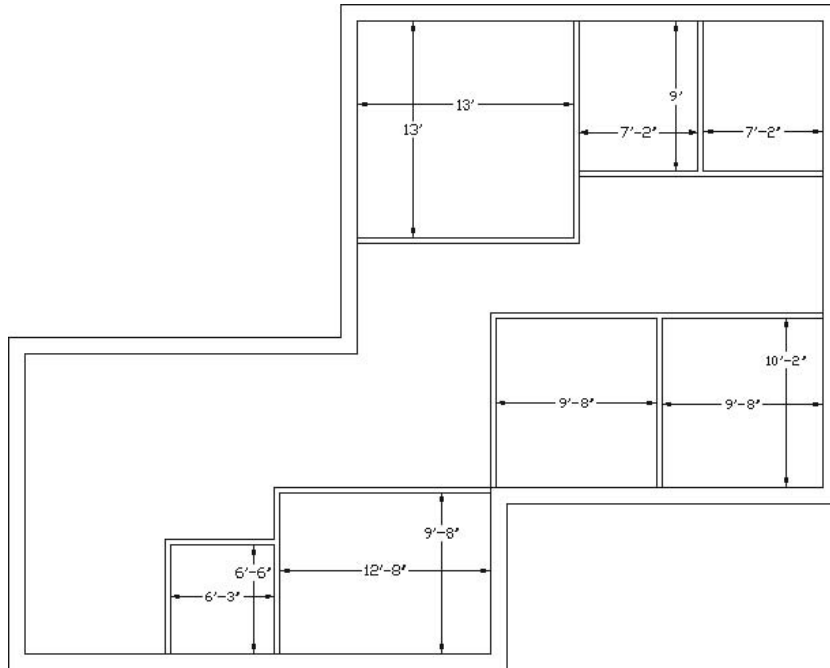


Workshop 2-B

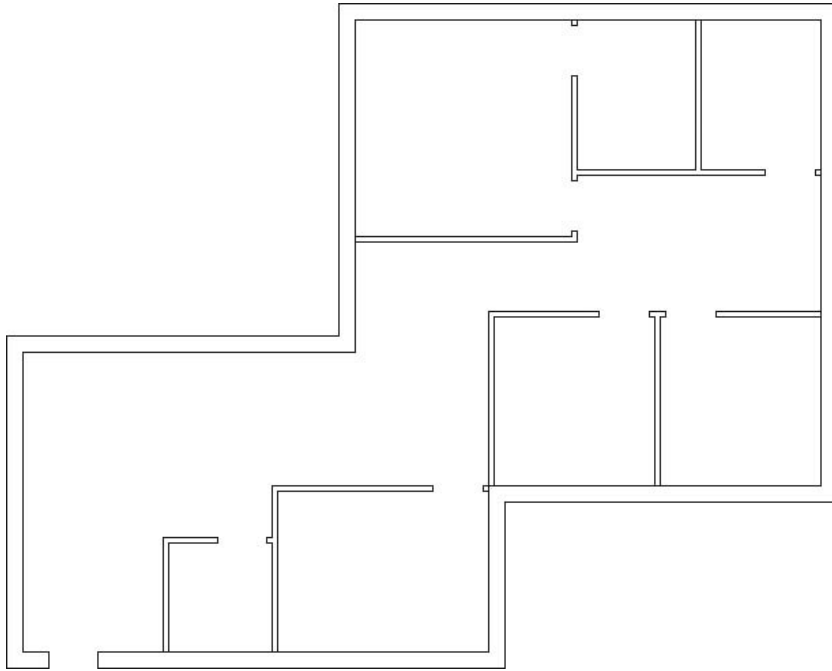
1. Start AutoCAD 2010 and open the file *Workshop_02.dwg*.
2. Make the layer **Walls** current.
3. Using the **Pline** command, draw the outer lines (without the dimensions) starting from point 16',10' using all the methods you learned in Chapter 2.



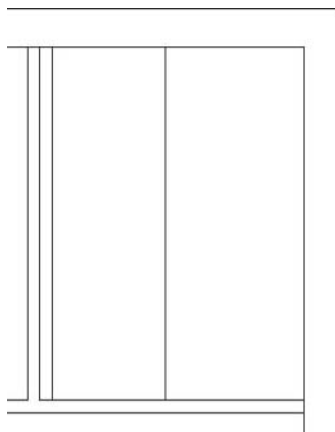
4. Using the **Offset** command, offset the polyline to the inside with an offset distance = 1'.
5. Explode the inner polyline.
6. Using the **Offset**, **Fillet**, **Chamfer**, **Trim**, **Extend**, **Lengthen**, and **Zoom** commands, create the interior walls using the following dimensions:



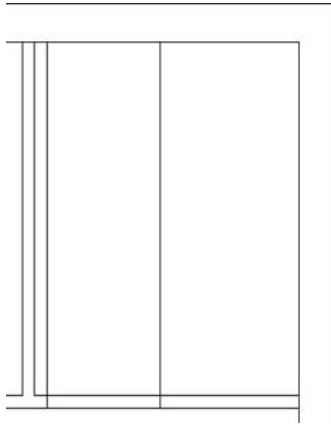
- The thickness of all inner walls = 4".
7. Make the door openings, taking the following into consideration:
 - a. All door openings = 3'.
 - b. Always take 4" clear distance from the walls for the door openings (except for the outside door take 1'6").



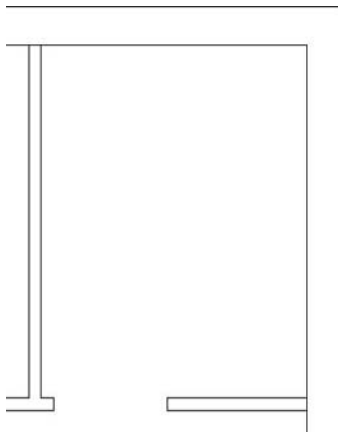
- To make the door openings, use the following technique:
 - Offset an existing wall (say, 4" for internal doors).
 - Offset the new line (say, 3' for room door).
 - You will have the following shape:



- Extend the two vertical lines to the lower horizontal line as shown in the following:



- Using the **Trim** command, select all horizontal and vertical lines as cutting edges, then press [Enter]. Trim the unnecessary parts by clicking on the objects (you can use **Crossing**, which is faster).
- This is what you will get:



8. Save the file and close it

CHAPTER REVIEW

1. You can use more than one offset distance within the same **Offset** command.
 - a. True, two offset distances can be used.
 - b. True, you can use as many as you wish.
 - c. False, only one offset distance can be used.
 - d. The only method available in the **Offset** command is the through point method.
2. In the **Lengthen** command, using the **Percent** option, 150% should be input as _____.
3. While you are trimming, you can extend, and vice versa.
 - a. True
 - b. False
4. You can fillet using an arc, but you need to specify:
 - a. Distances
 - b. Radius
 - c. Radius and distances
 - d. Length and angle
5. There are two methods to chamfer: Distances, and Length and Angle.
 - a. True
 - b. False
6. The first step in the **Extend** command is to select _____, and the second step is to select _____.
7. If you want to join lines, they should be _____, and _____ are allowed.

CHAPTER REVIEW ANSWERS

1. c
2. 150
3. a
4. b
5. a
6. Boundary Edge(s), Objects to extend
7. collinear, gaps

Chapter 5 MODIFYING COMMANDS

In This Chapter

- ◇ Introduction
- ◇ Selecting Objects
- ◇ The **Move** Command
- ◇ The **Copy** Command
- ◇ The **Rotate** Command
- ◇ The **Scale** Command
- ◇ The **Array** Command
- ◇ The **Mirror** Command
- ◇ The **Stretch** Command
- ◇ The **Break** Command
- ◇ **Grips**

5.1 INTRODUCTION

- In this chapter, we will learn the core of the modifying commands in AutoCAD®.
- We will cover nine commands, which will enable you to make any type of changes in a drawing.
- First, we will discuss the selection process (more in depth than what we discussed in Chapter 2).
- Then, we will discuss the following commands:
 - The **Move** command moves objects from one place to another.
 - The **Copy** command copies objects.
 - The **Rotate** command turns objects using rotation angles.
 - The **Scale** command creates larger or smaller objects using a scale factor.

- The **Array** command creates copies of objects in a matrix, circular, or semicircular fashion.
- The **Mirror** command creates mirror images of selected objects.
- The **Stretch** command either increases or decreases the length of objects.
- The **Break** command breaks an object into two pieces.
- We will wrap up with a discussion of **Grips** in AutoCAD.

5.2 SELECTING OBJECTS

- All of the modifying commands (with some exceptions) will ask you the same question:

Select objects:

- In Chapter 2, we looked at some of the methods used to select objects. We will now expand our knowledge in this area.
- All of the methods we discuss will involve typing at least one letter in the **Command Window** at the **Select objects** prompt.

W (Window)

- If you typed **W**, the **Window** mode will be available whether you moved to the right or to the left.

C (Crossing)

- If you typed **C**, the **Crossing** mode will be available whether you moved to the right or to the left.

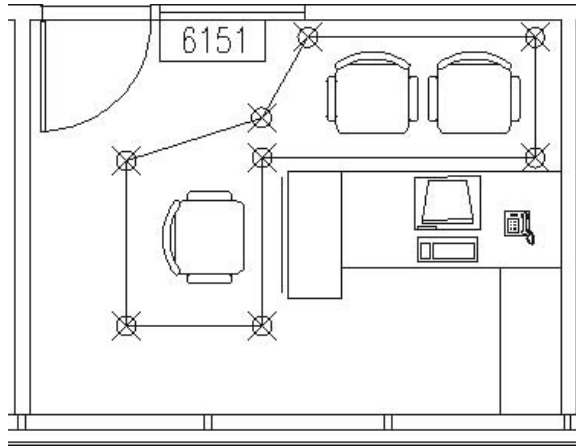
WP (Window Polygon)

- If you want to select multiple objects without the constraint of the rectangle window, you can use the **Window Polygon** mode.
- When you type **WP** and press [Enter], the following prompt will appear:

First polygon point: (*specify the first point of the polygon*)
Specify endpoint of line or [Undo]: (*specifying the second point*)
Specify endpoint of line or [Undo]: (*specify the third point, etc.*)

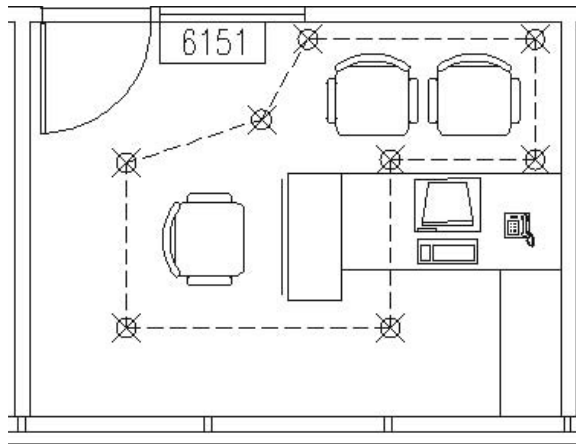
- When you are done, press [Enter] to end the **WP** mode.

- The objects that are fully inside the **Window Polygon** will be selected. If any part (even a small part) is outside the shape, it will not be selected. See the following example:



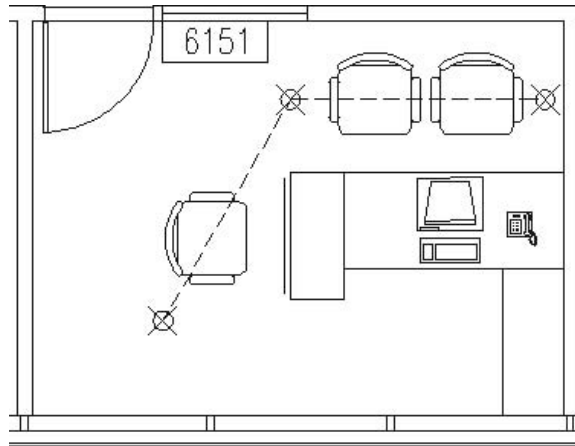
CP (Crossing Polygon)

- The **Crossing Polygon** mode is the same as the **Window Polygon** mode, except it has the features of the **Crossing** mode. That means that whatever it fully contains—plus any object that touches it—will be selected.



F (Fence)

- The main function of the **Fence** mode is to touch objects.
- The **Fence** mode was discussed when we introduced the **Trim** and **Extend** modes.



L (Last)

- The **Last** mode is used to select the last object drawn.

P (Previous)

- The **Previous** mode is used to select the last selection set used.

All

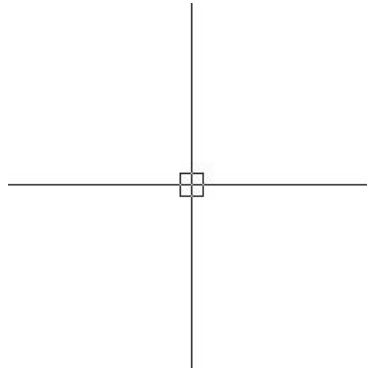
- To select all objects in the current file use **All**.

Deselect

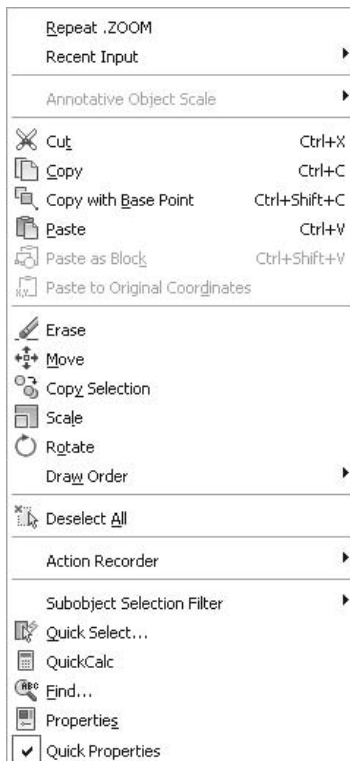
- If you select a group of objects, then you discover that one or two of the objects were selected by mistake. How do you deselect them?
- Simply hold the [Shift] key, click these objects, and they will be deselected.

Other Methods for Selecting Objects

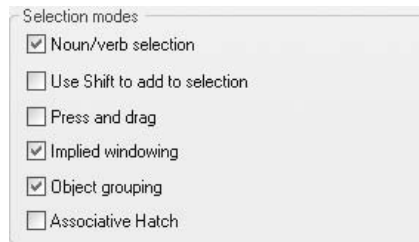
- There are other methods used to select objects that will make your life easier.
- There is a good technique called the **Noun/Verb selection**, which will allow the user to select an object first and then issue the command.
- The cursor looks like the following:



- As you can see, there is a pick box inside the cursor.
- Without issuing any command, you can:
 - Click on any object to select it.
 - Find an empty space, click, and then go to the right to get the **Window** mode.
 - Find an empty space, click, and then go to the left to get the **Crossing** mode.
- Once you select the desired objects, right-click to get the following shortcut menu:



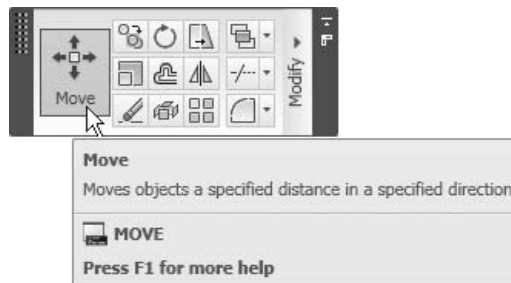
- From this shortcut menu, you can access five modifying commands without typing a single letter on the keyboard or issuing any commands from the menus or the toolbars. These commands are: **Erase**, **Move**, **Copy Selection**, **Scale**, and **Rotate**.
- Make sure that the **Noun/Verb selection** mode is turned on using the **Menu Browser** by selecting **Tools/Options**, then the **Selection** tab, and under **Selection modes**, click the **Noun/Verb selection** checkbox to turn it on (that is, if it was off).



- This technique will not work with the **Offset**, **Fillet**, **Chamfer**, **Trim**, **Extend**, or **Lengthen** commands. It will work with the **Join** command.

5.3 THE MOVE COMMAND

- The **Move** command is used to move objects from one place to another.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Move** button.



- The **Move** command is a three-step command.
- The first step is to:

Select objects:

- Once you are done, press [Enter] or right-click.

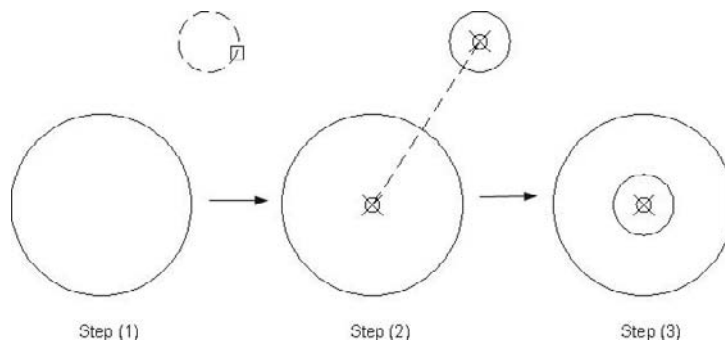
- The next prompt will ask you to:

Specify base point or [Displacement]
<Displacement>: (*Specify the base point*)

- The *base point* will be used in four other commands. What is a base point?
 - The simplest way to define a base point is to call it a *handle* point.
 - There is no golden rule that defines a right point as a base point.
 - Rather, you have to take it case by case, so it may be the center of a group of objects or it may be in the upper left-hand corner.
 - This is also true for commands like **Move**, **Copy**, and **Stretch**. But for a command such as **Rotate**, the base point is the point where the whole shape will rotate around. In the **Scale** command, it will be the point that the whole shape will shrink or enlarge relative to it.
- The third prompt will be:

Specify second point or <use first point as displacement>: (*Specify the second point*)

- The command will end automatically.
- See the following example:

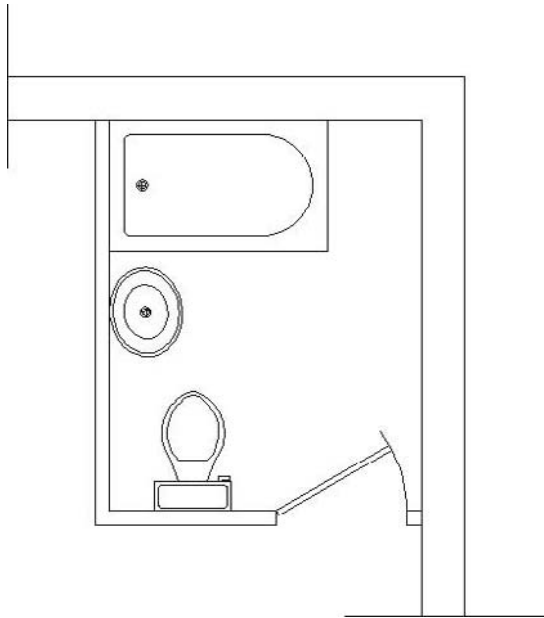


MOVING OBJECTS



Exercise 23

1. Start AutoCAD 2010.
2. Open the file *Exercise_23.dwg*.
3. Move the four objects (bathtub, toilet, sink, and door) to their respective places to make the bathroom look like the following:



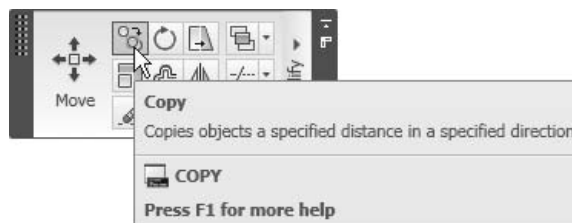
4. Save the file and close it.



- For the toilet, use the midpoint of the inside wall.
- For the sink, use the left quadrant and the midpoint of the inside wall.

5.4 THE COPY COMMAND

- The **Copy** command is used to copy objects.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Copy** button.



- The **Copy** command is a three-step command.
- The first step is to:

Select objects:

- Once you are done, press [Enter] or right-click.
- The next prompt will ask you to:

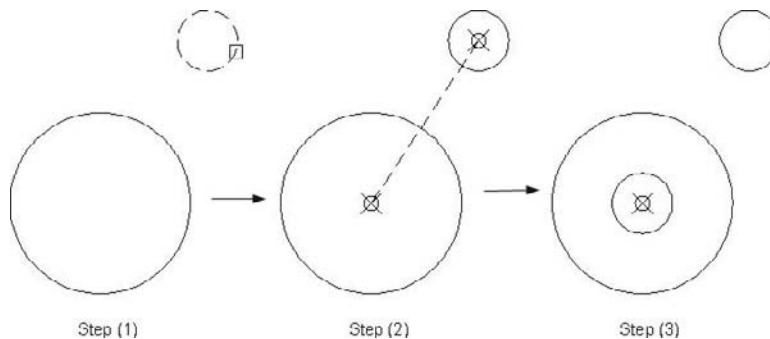
Specify base point or [Displacement] <Displacement>:
(Specify the base point)

- The third prompt will be to:

Specify second point or [Exit/Undo] <Exit>: **(Specify the second point)**

Specify second point or [Exit/Undo] <Exit>: **(Specify another second point)**

- Once you are done, press [Enter] or right-click.
- If you made a mistake, simply type **u** into the **Command Window** to undo the last action.
- See the following example:

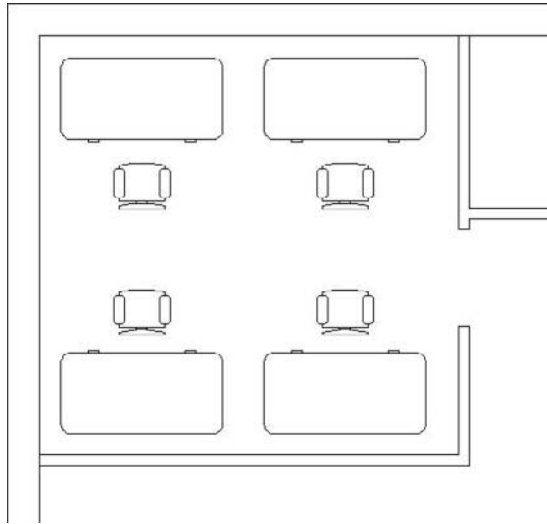


COPYING OBJECTS



Exercise 24

1. Start AutoCAD 2010.
2. Open the file *Exercise_24.dwg*.
3. Copy the chair three times to make the room look like the following:



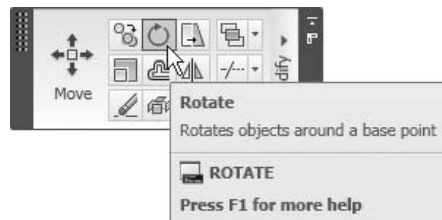
4. Save the file and close it.



- Use **Polar Tracking**, **OSNAP**, and **OTRACK** to make sure they are aligned.

5.5 THE ROTATE COMMAND

- The **Rotate** command is used to rotate objects around a point using a rotation angle.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Rotate** button.



- The **Rotate** command is a three-step command.
- The first step is to:

Select objects:

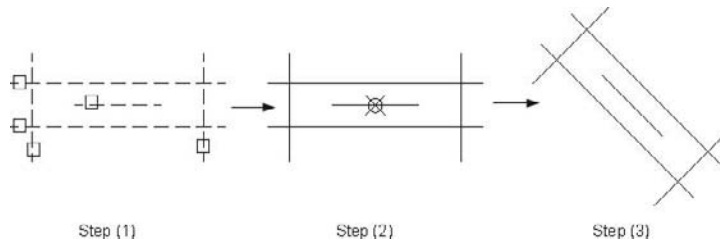
- Once you are done, press [Enter] or right-click.
- The next prompt will ask you to:

Specify base point: (*Specify the base point*)

- The third prompt will be to:

Specify rotation angle or [Copy/Reference] <0>:
(*specify the rotation angle, -=CW, +=CCW*)

- You can use the **Copy** option if you want to rotate a copy of the objects selected while keeping the original intact.
- The command will end automatically.
- See the following example:

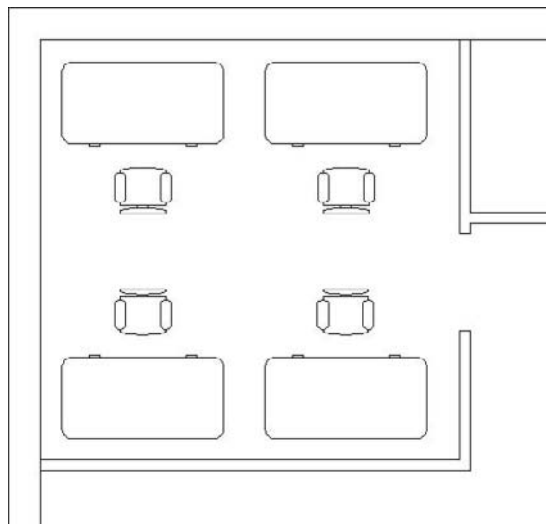


ROTATING OBJECTS



Exercise 25

1. Start AutoCAD 2010.
2. Open the file *Exercise_25.dwg*.
3. Rotate the two chairs (each using a different command) around the center of each chair to make the room look like the following:



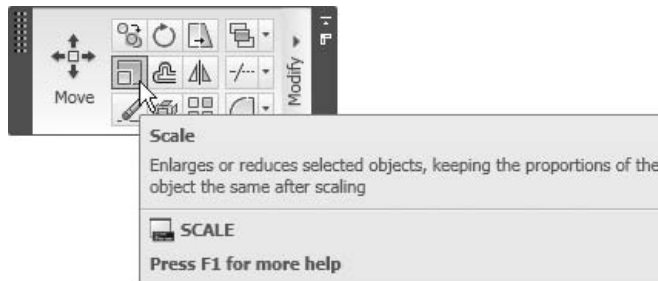
4. Save the file and close it.



- The best way to do this exercise is to use either **Polar Tracking** or type the angles. You may need to move some things around to make the room look perfect.

5.6 THE SCALE COMMAND

- The **Scale** Command is used to enlarge or shrink objects using a scale factor.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Scale** button.



- The **Scale** command is a three-step command.
- The first step is to:

Select objects:

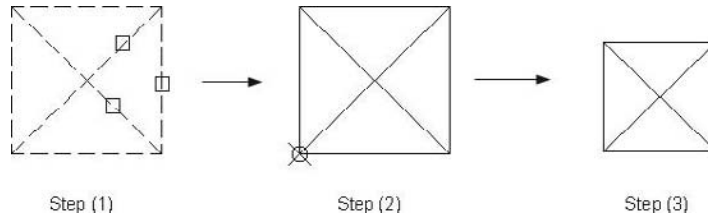
- Once you are done, press [Enter] or right-click.
- The next prompt will ask you to:

Specify base point: (*Specify the base point*)

- The third prompt will be to:

Specify scale factor or [Copy/Reference] <1.0000>:
(*specify the scale factor, the number should be a non-zero positive number*)

- You can use the **Copy** option if you want to scale a copy of the objects selected while keeping the original intact.
- The command will end automatically.
- See the following example:

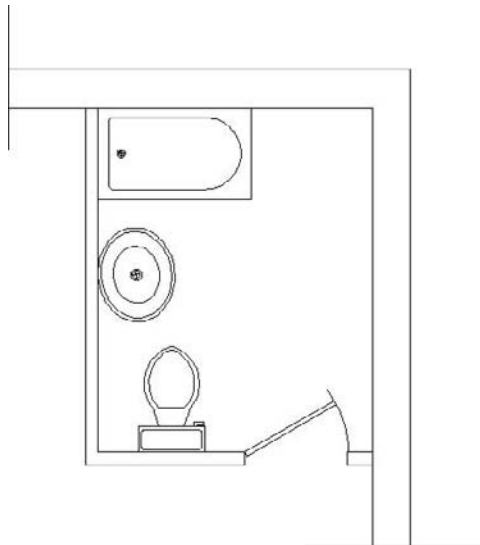


SCALING OBJECTS



Exercise 26

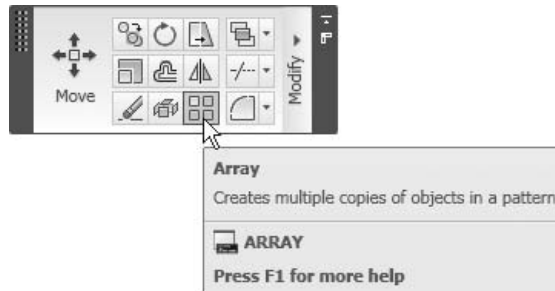
1. Start AutoCAD 2010.
2. Open the file *Exercise_26.dwg*.
3. Set the scale factor = 0.8 to scale the bathtub using the upper left-hand corner as the base point.
4. Set the scale factor = 1.2 to scale the sink using the quadrant on the left side as the base point.
5. The room should look like the following:



6. Save the file and close it.

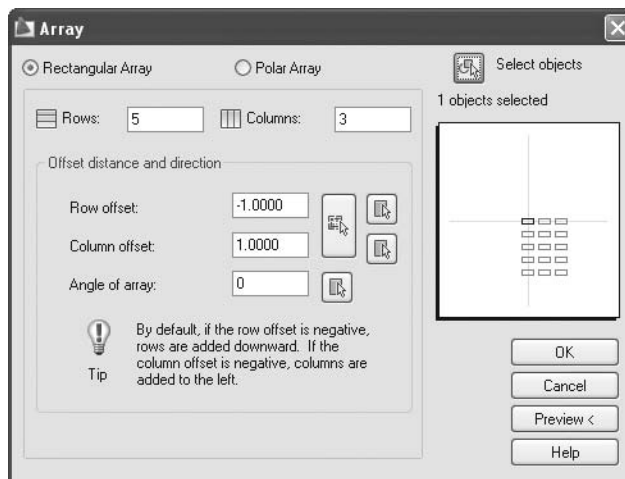
5.7 THE ARRAY COMMAND

- The **Array** command is used to create duplicates of objects using two methods:
 - Rectangular array (matrix shape)
 - Polar array (circular or semi circular shape)
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Array** button.



Rectangular

- If you want to create a duplicate of objects simulating the matrix shape, you should select the **Rectangular Array**.
- The following dialog box will appear:

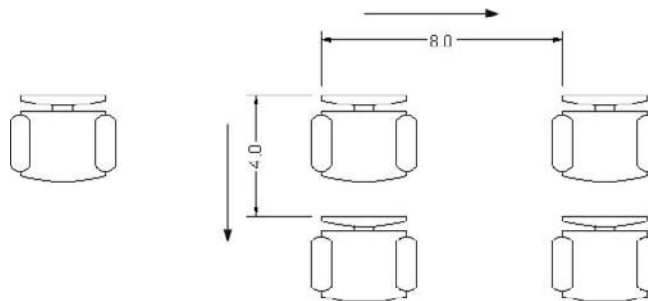


- First, click the **Select Objects** button to select the desired objects. Once you are done, press [Enter] or right-click.

- Next, specify the number of **Rows** and the number of **Columns** (the original object is inclusive).
- Specify the **Row offset** (the distance between rows), and specify the **Column offset** (the distance between columns). While you are doing this keep two things in mind:
 - You have to be consistent. Measure the distance from the same reference point (e.g., top-to-top, bottom-to-bottom, center-to-center, etc.).
 - You have to take note of the direction in which you are copying. If you input a positive number, it will be either to the right or up. If you input a negative number, it will be either to the left or down.
- Specify the **Angle of array**. By default, it will repeat the objects using the orthogonal angles.
- Click the **Preview** button to see the result of your input.
- AutoCAD will display the result and the following prompt will appear:

Pick or press Esc to return to dialog or <Right-click to accept array>:

- If you like the result, press [Enter] or right-click.
- If not, press [Esc].
- See the following example:



Step (1)

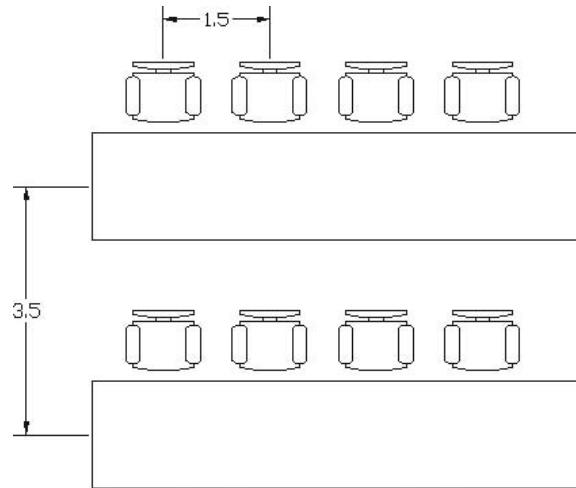
Step (2)

RECTANGULAR ARRAY



Exercise 27

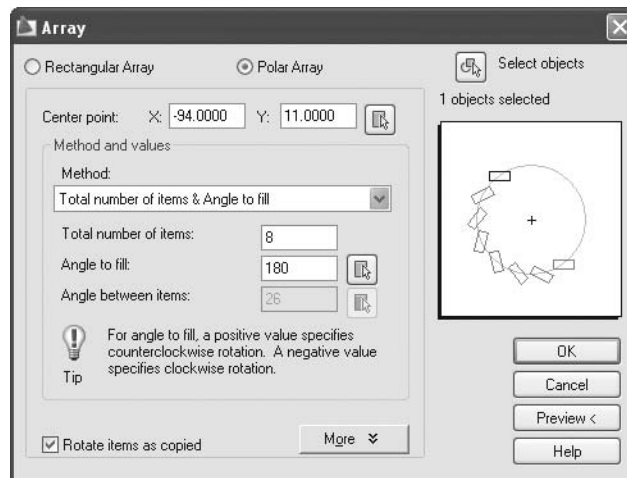
1. Start AutoCAD 2010.
2. Open the file *Exercise_27.dwg*.
3. Using the **Rectangular Array**, array the chairs to look like the following:



4. Save the file and close it.

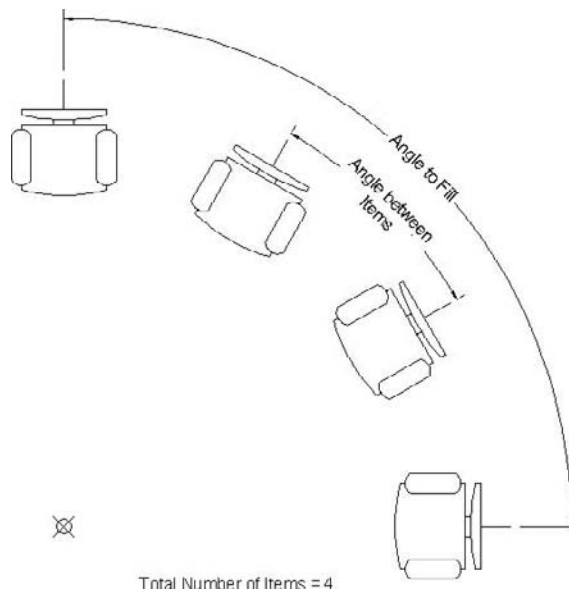
The Polar Array Command

- The **Polar Array** command is used if you want to duplicate certain objects simulating circular or semicircular shapes.
- The following dialog box will appear:

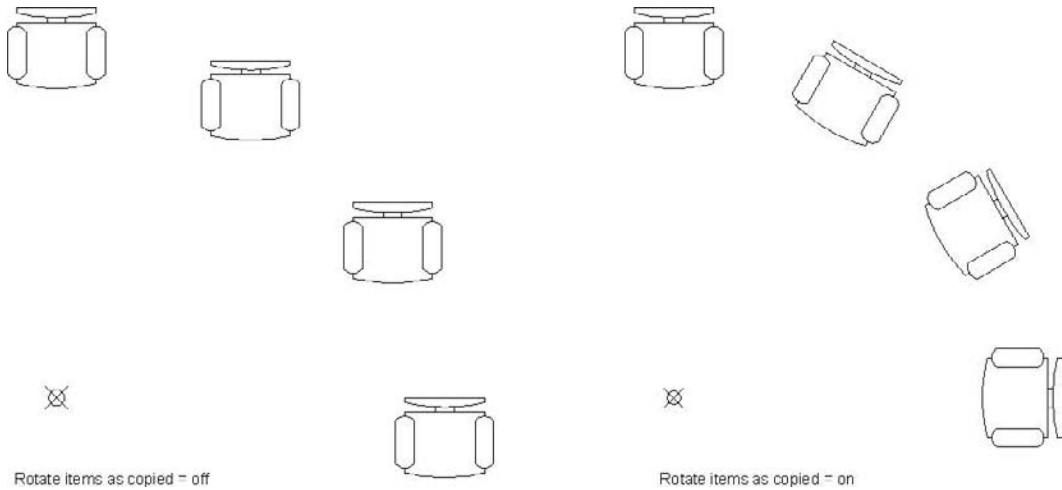


- First, click the **Select Objects** button to select the desired objects. Once you are done, press [Enter] or right-click.

- Next, specify the **Center point** of the array, either by inputting the coordinates in X and Y, or by clicking the **Pick Center Point** button and specifying the point by using the mouse.
- You have three pieces of data to input and AutoCAD will only take two of them. These are:
 - Total number of items
 - Angle to fill
 - Angle between items
- The following diagram illustrates the relationship between the three parameters:



- You can specify two out of three parameters, which gives you three different methods. They are:
 - Method 1: Specify the total number of items and the angle to fill. AutoCAD will figure out the angles between the items.
 - Method 2: Specify the total number of items and the angle between the items. AutoCAD will know the angle to fill.
 - Method 3: Specify the angle to fill and the angle between the items, and AutoCAD will calculate the total number of items.
- Under **Methods and values**, select the proper method and input the corresponding values.
- Specify whether or not you want to **Rotate items as copied**. See the following example:

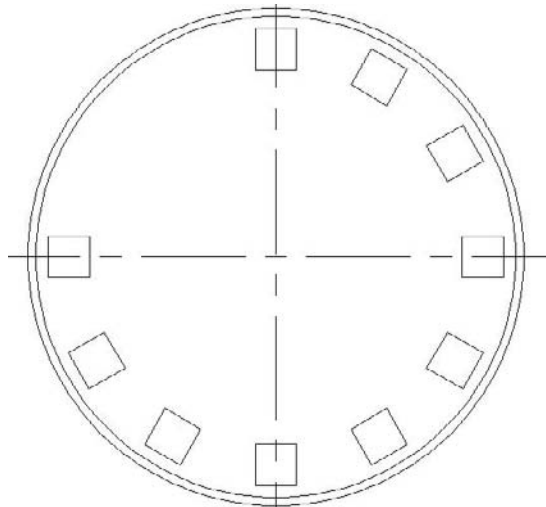


POLAR ARRAY



Exercise 28

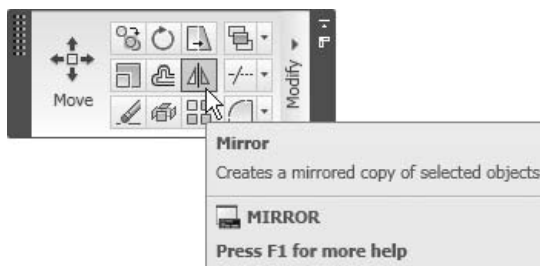
1. Start AutoCAD 2010.
2. Open the file *Exercise_28.dwg*.
3. Using the **Polar Array** command, array the square so you will get the following result:



4. Save the file and close it.

5.8 THE MIRROR COMMAND

- The **Mirror** command is used to create a mirror image of selected objects.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Mirror** button.



- The first step is to:

Select objects:

- Once you are done, press [Enter] or right-click.
- Now, you need to specify the mirror line by specifying two points:

Specify first point of mirror line: (*specify the first point of the mirror line*)

Specify second point of mirror line: (*specify the second point of the mirror line*)

- The following applies to the mirror line:
 - There is no need to draw a line to act as a mirror line; two points will do the job.
 - The length of the mirror is not important, but the location and angle of the mirror line will affect the final result.
- The last prompt will be:

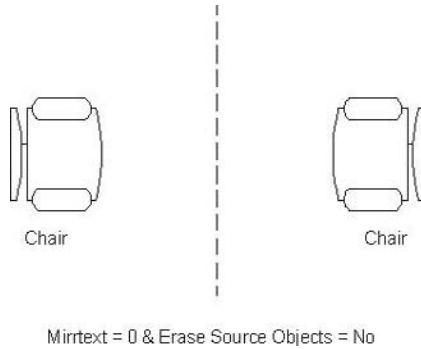
Erase source objects? [Yes/No] <N>: (*type N, or Y*)

- The **Mirror** command will produce an image in all cases, but what should AutoCAD do with the source objects? You can keep them or erase them.
- The **Mirror** command ends automatically.
- If part of the objects to be mirrored is text, you have to control whether you want to treat it as any other object and mirror it, or simply copy it.
- To do that, prior to issuing the **Mirror** command, type **mirrtext** in the **Command Window** and the following prompt will appear:

Enter new value for MIRRTEXT <0>:



- If you input 0 (zero), then the text will be copied.
- If you input 1, then the text will be mirrored.
- See the following example:

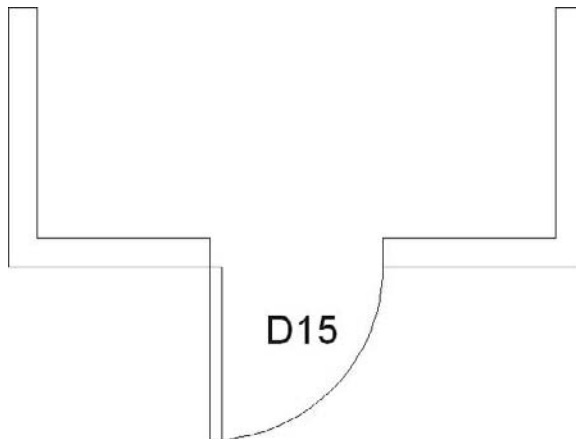


MIRRORING OBJECTS



Exercise 29

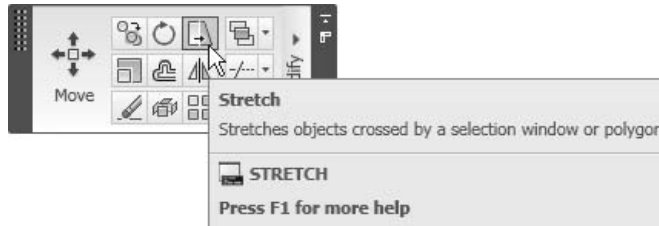
1. Start AutoCAD 2010.
2. Open the file *Exercise_29.dwg*.
3. Using the **Mirror** command, create the following shape:



4. Save the file and close it.

5.9 THE STRETCH COMMAND

- The **Stretch** command is used to increase or decrease the length of selected objects.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Stretch** button.



- The first step is to:

Select objects to stretch by crossing-window or crossing-polygon...

- The **Stretch** command is one of the few commands that insists on a certain method of selection.
- The **Stretch** command asks users to select using either **C** or **CP**.
- As we discussed previously, **C** and **CP** will select any object contained inside and any object touched (crossed) by **C** or **CP** lines.
- The **Stretch** command will utilize both facilities by setting the following rules:
 - Any object contained fully inside **C** or **CP** will be moved.
 - Any object crossed by **C** or **CP** will be stretched.
- Once you are done, press [Enter] or right-click.
- The second prompt will be to:

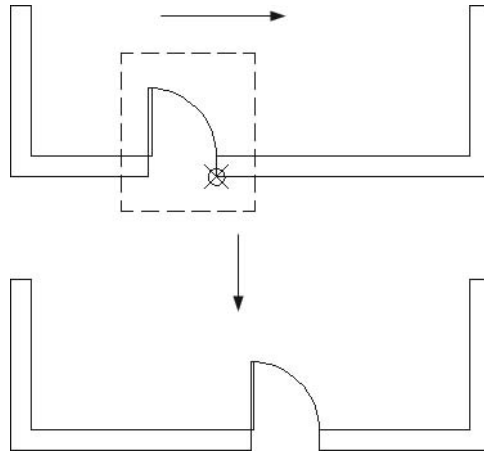
Specify base point or [Displacement]
<Displacement>: (*specify Base point*)

- The third prompt will be to:

Specify second point or <use first point as displacement>:
(*specify the destination point*)

- The **Stretch** command ends automatically.

- See the following example:

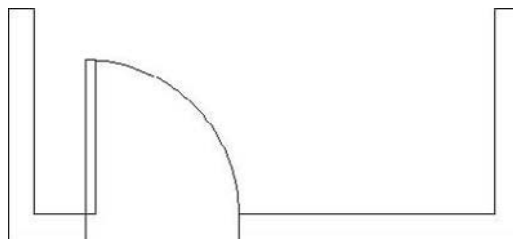


STRETCHING OBJECTS



Exercise 30

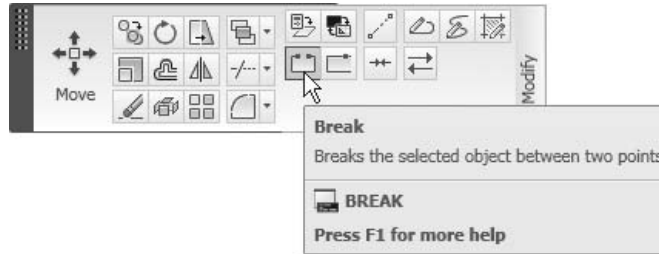
1. Start AutoCAD 2010.
2. Open the file *Exercise_30.dwg*.
3. Using the **Stretch** command, stretch the door 2 units to the left so it will look like the following:



4. Save the file and close it.

5.10 THE BREAK COMMAND

- The **Break** command is used to break an object into two pieces.
- Make sure you are in the **Home** tab on the **Ribbon**, and, by using the **Modify** panel, expand it and select the **Break** button.



- The first step is to:

Select object:

- You can break one object at a time. When you select this object, AutoCAD will prompt you with the following:

Specify second break point or [First point]:

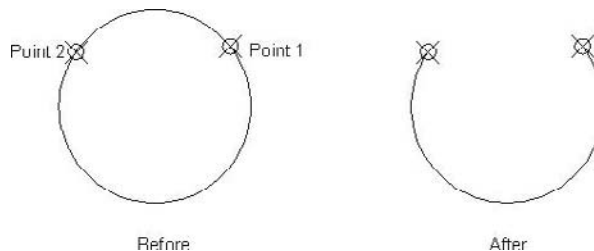
- To understand this prompt, take note of the following points:
 - In order to break an object you have to specify two points on it.
 - The selection you make can be considered either a selection and a first point or a selection only. If you consider the selection as a selection and first point, respond to this prompt by specifying the second point.
 - If you want the selection to only be the selection, type the letter **F** in the **Command Window** and AutoCAD will respond with the following prompt:

Specify first break point: (*specify first breaking point*)

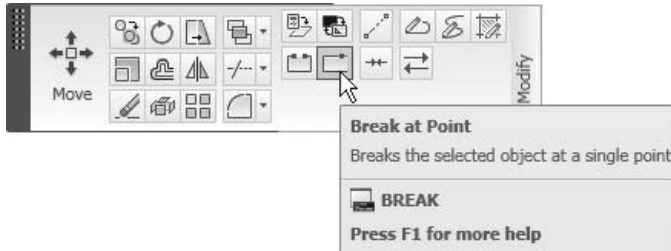
Specify second break point: (*specify second breaking point*)



- When you want to break a circle, take care to specify the two points CCW.
- See the following example:



- There is another tool called **Break at Point** in the **Modify** panel on the **Home** tab.



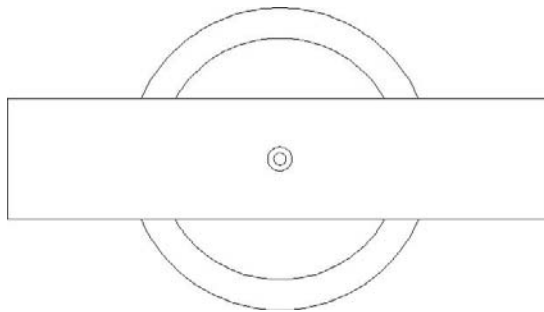
- This is similar to the **Break** command except for the following differences:
 - You will be asked to select only one point.
 - AutoCAD will assume that the first point and the second point are in the same place.
 - The object will be broken into two objects but connected.

BREAKING OBJECTS



Exercise 31

1. Start AutoCAD 2010.
2. Open the file *Exercise_31.dwg*.
3. Using the **Break** command, break the two circles to look something like this:

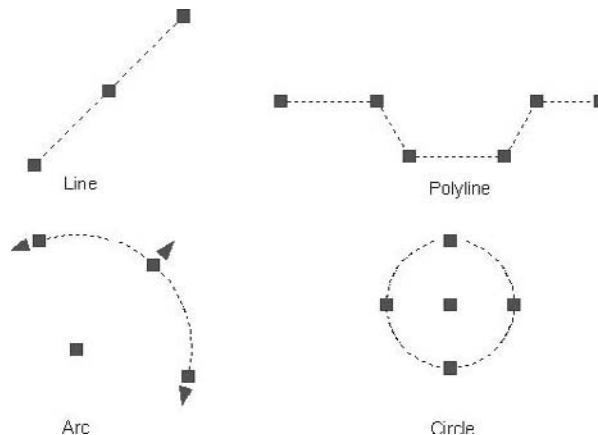


4. Save the file and close it.

5.11 GRIPS

Introduction

- **Grips** is a method of modifying your objects easily and quickly.
- **Grips** is done with a simple click on an object, or multiple objects, without issuing any commands.
- **Grips** will do two things for you:
 - It will select the objects, and they will be ready for any modifying command to be issued, as they will act as a selection set.
 - Blue (default color) squares will appear at certain places depending on the type of the object. Here are some examples:

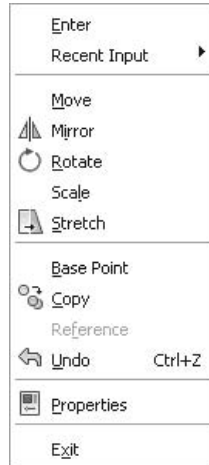


- These squares are the grips.
- There is a magnet relationship between these squares and the pick box of the crosshairs.
- Also, if you hover over grips, the blue will turn to green, indicating the current grip.
- If you click on one of these blue squares, you will:
 - Make it hot, turning it to red
 - Make this grip a base point
 - Start a group of five **Modify** commands (using the right-click)

The Five Commands

- Once you make one of the blue squares hot (by clicking on it), this grip will become the base point for five commands, they are:
 - **Move**
 - **Mirror**

- **Rotate**
 - **Scale**
 - **Stretch**
- To see these commands, right-click and the following shortcut menu will appear:



- The other options available in the shortcut menu are:
- **Base Point**, which is used to define a new base point other than the one you started with.
 - **Copy**, which is a mode rather than a command. **Copy** mode works with the other five commands and will give you the ability to **Rotate with Copy**, **Scale with Copy**, etc.

Steps

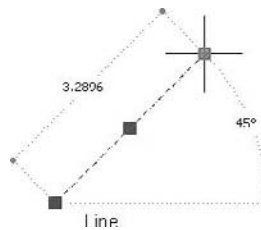
- The steps to use **Grips** are as follows:
- Select the object(s) desired (direct clicking, **Window** mode, or **Crossing** mode).
 - Select one of the grips to be your base point and click it. It will become hot (red by default).
 - Right-click and select the desired command from the shortcut menu. You can now specify another base point and/or you can select the **Copy** mode.
 - Perform the steps of the desired command.
 - Once you are done, press [Esc] either once or twice depending on the command you are working with.



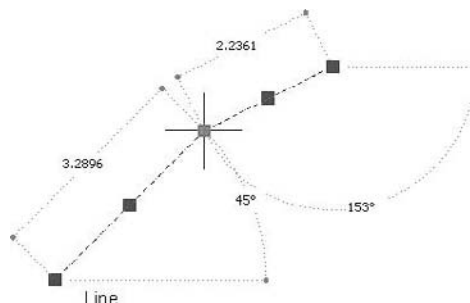
- You can use **OSNAP** with **Grips** with no limitations. Also, you can use **Polar Tracking** and **OTRACK**, so the modification will be accurate.
- **Mirror** is the only command that does not explicitly ask for a base point. Why is it listed with the other four commands? The answer is that AutoCAD considers the first point of the mirror line to be the base point.
- In order to keep both the original and the mirrored image using **Grips**, you have to select the **Copy** mode after you select the **Mirror** command.
- You can deselect certain objects from the grips by holding the [Shift] key and clicking on the object, avoiding the blue squares.

Dynamic Input

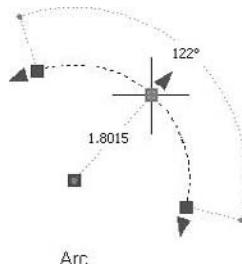
- **Dynamic Input** can give you information about the objects with their grips appearing on the screen.
- If you hover over an end grip of a line, **Dynamic Input** tells you the length and the angle of that line.



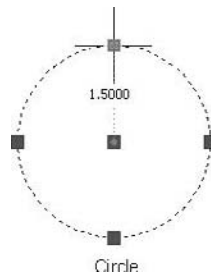
- If you hover over an end grip shared between two lines, **Dynamic Input** tells you the length and angle of both lines:



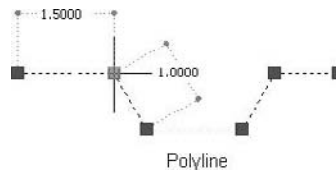
- If you hover over the middle grip of an arc, **Dynamic Input** tells you the radius and the included angle of the arc:



- If you hover over the quadrant grip of the circle, **Dynamic Input** tells you the radius of the circle:



- If you hover over an intermediate grip of a polyline, **Dynamic Input** tells you the lengths of the line segments (without the angles):



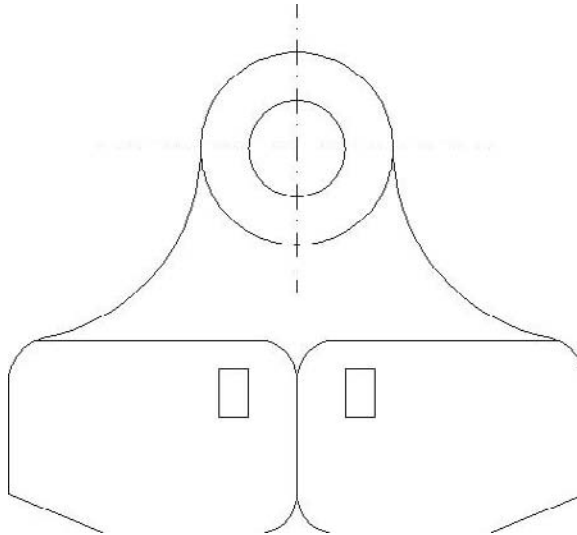
USING GRIPS



Exercise 32

1. Start AutoCAD 2010.
2. Open the file *Exercise_32.dwg*.
3. Without issuing any commands, select the upper circle. Make the center hot, then right-click and select **Scale**, then right-click again and select **Copy**. At the **Scale factor** prompt type **0.5**. Press [Esc] twice.
4. In the right part of the base, without issuing any commands, select the rectangle. Make one of the blue grips hot by clicking it, right-click, select **Rotate**, then right-click again and select **Base Point**. Specify a new base point, which is the center of the rectangle (using **OSNAP** and **OTRACK**), and setting the Rotation angle = 90. Press [Esc] twice.

5. Select the rotated rectangle at the right part of the base, select any grip to make it hot, then right-click and select **Mirror**. Right-click again and select **Copy**. Right-click for a third time and select **Base Point**. Specify one of the two endpoints of the vertical lines separating the two parts of the base and then specify the other endpoint. Press [Esc] twice. The shape will look like this:



6. Save the file and close it.

CHAPTER REVIEW

1. What do the commands **Move**, **Copy**, **Rotate**, **Scale**, and **Stretch** have in common?
 - a. They are all modifying commands.
 - b. They all use the base point concept.
 - c. They all change the length of an object.
 - d. A and B.
2. In the **Stretch** command, you have to use _____ or _____ while selecting objects.
3. **Mirrtext** is used to control whether to copy or mirror the text in the **Mirror** command.
 - a. True
 - b. False

4. If you break a circle, take care to specify the two points:
 - a. CCW
 - b. CW
 - c. It doesn't matter
 - d. You can't break a circle
5. You can scale using a Scale factor = -1.
 - a. True
 - b. False
6. In the **Array** command and the **Rectangular** option, the **Row offset** must be _____ if you want to repeat the objects downward.

CHAPTER REVIEW ANSWERS

1. d
2. **C** or **CP**
3. a
4. a
5. b
6. negative

Chapter 6 DEALING WITH BLOCKS

In This Chapter

- ◇ What Are Blocks?
- ◇ Creating Blocks
- ◇ Inserting Blocks
- ◇ Exploding Blocks
- ◇ Using **Design Center**
- ◇ Automatic Scaling
- ◇ What Is a Tool Palette?
- ◇ Creating a Tool Palette
- ◇ Customizing a Tool Palette
- ◇ Editing Blocks

6.1 WHAT ARE BLOCKS?

- A block in AutoCAD® is any shape that is repeated in one or more drawings more than once.
- Instead of drawing it each and every time you need it, use the following steps:
 - Draw it once.
 - Store it as a block.
 - Insert it as many times as you wish.
- Blocks in AutoCAD have changed over the years, which has made some of the old procedures obsolete.
- In this chapter, we will discuss the old methods, but we will concentrate more on the new methods of using blocks.

6.2 CREATING BLOCKS

- The first step in creating blocks in AutoCAD is to draw the desired shape.
- While drawing the shape, consider the following three guidelines:
 - Draw the shape in Layer 0 (zero).
 - Draw the shape in certain units.
 - Draw the shape in the right dimensions.

Why Layer 0?

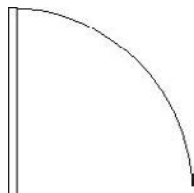
- Layer 0 is different from any other layer in AutoCAD, because it will allow the block to be transparent both in color and in linetype.
- If you draw the shape that will be a block while layer 0 is current, then insert it into another layer with red color and dashdot linetype so the block will be red and dashdot.

Why Certain Units?

- If you want AutoCAD to automatically rescale your block to fit into the current drawing units, you have to specify the units of the block.

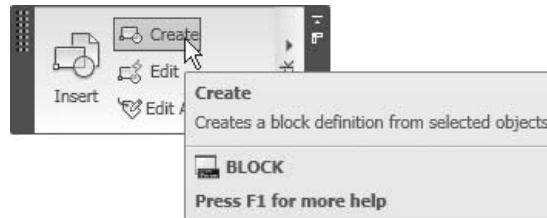
What Are the Right Dimensions?

- Right dimensions are either:
 - The real dimensions of the shape.
 - Values for the distances, such as 1, 10, 100, 1000. It will be easier to scale the block once you insert it.
- Let's assume we draw the following shape:

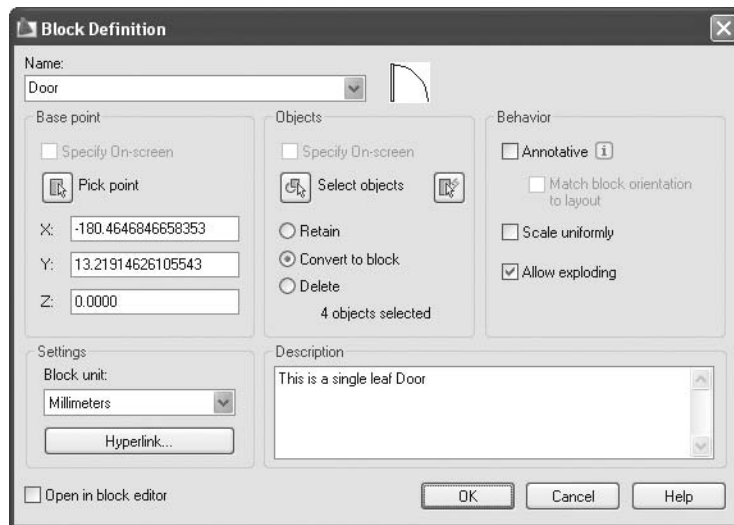


- The next step would be to think about a point that will act as the base point (the handle for this block).
- Also, think of a good name for this block.
- With all of these in mind, you can now issue the command.

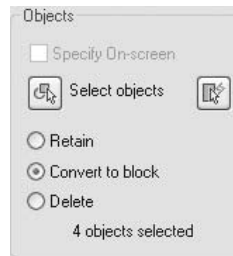
- Make sure you are in the **Home** tab on the **Ribbon**, and, by using the **Block** panel, select the **Create** button:



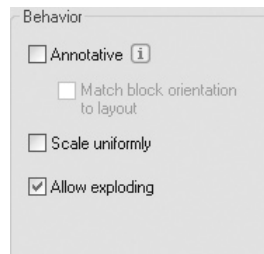
- The following dialog box will appear:



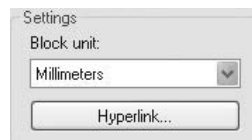
- Type the name of the block (similar to the layer naming conditions, it should not exceed 255 characters).
- Under **Base point**, click the **Pick point** button to input the base point of the block. Once you are done, press [Enter] or right-click. Or, you can select the checkbox **Specify On-screen** to specify the base point after the dialog box is closed.
- Under **Objects**, click the **Select objects** button to select the objects. Once you are done, press [Enter] or right-click. Or, you can select the checkbox **Specify On-screen** to specify the base point after the dialog box is closed.
- To decide what to do with the objects you draw to create this block, select one of the following three choices:
 - Retain
 - Convert to block
 - Delete



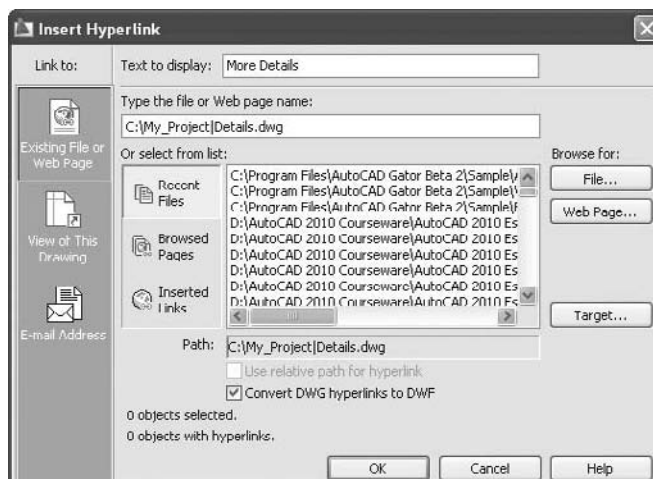
- Under **Behavior**, set the following:
 - Leave **Annotative** off (this is an advanced feature).
 - Choose to **Scale uniformly** (X-scale = Y-scale) or not.
 - Choose whether to **Allow exploding** or not.



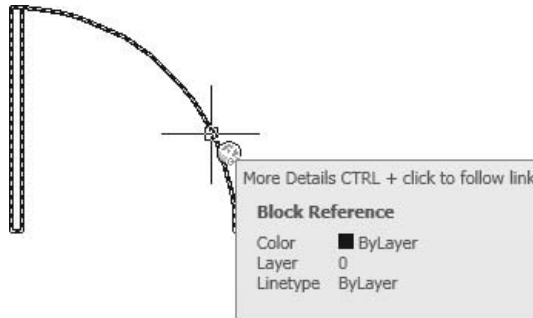
- Under **Settings**, select the **Block unit** you will be using in your drawing. This will help AutoCAD in the **Automatic scaling** feature.



- Click the **Hyperlink** button and the following dialog box will appear:



- This dialog box will allow users to insert a hyperlink inside the block to take them to a website, a drawing file with more details, an MS Word file, an Excel file for calculation, etc.
- After you finish, when you approach the block, you will see something like the following:



- As you can see, the shape of the hyperlink will be displayed and a help message will appear telling you to hold the [Ctrl] key and click the block to open the desired link.
- Write any description for your block.
- Select whether to allow this block to be opened in the block editor (block editor is an advanced feature used to create dynamic blocks) or to keep it off for the time being.
- When you are done, click **OK**.
- Now, let's imagine that our drawing has a cabinet, the door of the cabinet will be opened, and the defined block will be put inside it. This block will be intact. Even when you insert it, you will only insert a copy of it.
- You can define as many blocks as you wish.

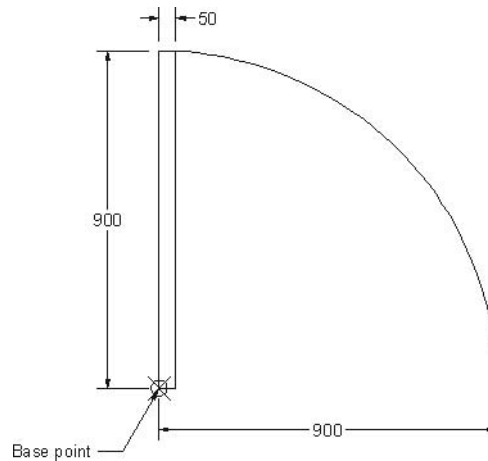
CREATING A BLOCK (METRIC)



Workshop 3-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_03.dwg*.
3. Make layer 0 current.

4. Choose an empty space and draw the following shape (without dimensions):



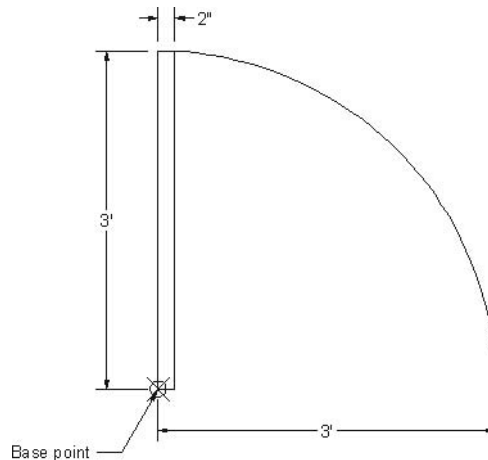
5. Using the **Make Block** command, create a block using the following information:
- Block name = Door
 - Specify the designated base point.
 - Delete the shape after the creation of the block.
 - Block unit = Millimeters
 - Scale uniformly = off, Allow exploding = on
 - Description = Door to be used inside the building. Refer to the door table.
 - Open in block editor = off
6. Save the file and close it.

CREATING A BLOCK (IMPERIAL)



Workshop 3-B

- Start AutoCAD 2010.
- Open the file *Workshop_03.dwg*.
- Make layer 0 current.
- Choose an empty space and draw the following shape (without dimensions):

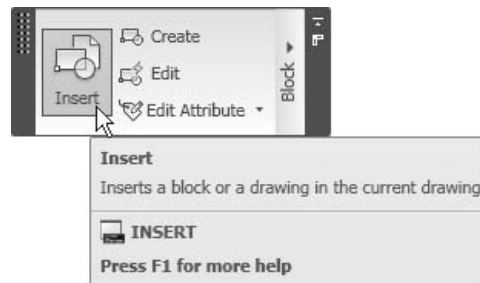


5. Using the **Make Block** command, create a block using the following information:
 - a. Block name = Door
 - b. Specify the designated base point.
 - c. Delete the shape after the creation of the block.
 - d. Block unit = Inches
 - e. Scale uniformly = off, Allow exploding = on
 - f. Description = Door to be used inside the building. Refer to the door table.
 - g. Open in block editor = off
6. Save the file and close it.

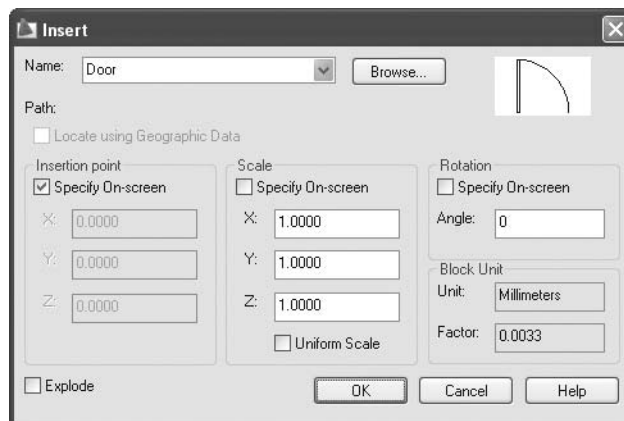
6.3 INSERTING BLOCKS

- Once you create a block, you can use it in your drawing as many times as you wish.
- When inserting a block into your drawing, consider the following guidelines:
 - Set the desired layer to be current.
 - Prepare the drawing to accommodate the block (e.g., finish the door openings before inserting the door block).
- After you consider all of these guidelines, issue the command.

- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Block** panel, select the **Insert** button.

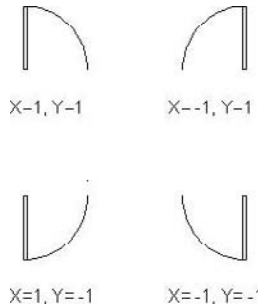


- The following dialog box will appear:



- Select the name of the desired block from the list.
- Specify the **Insertion point**, using one of two methods:
 - Click the **Specify On-screen** checkbox. By doing so, you will specify the insertion point using the mouse (this is easier than the next method).
 - Type the coordinates of the insertion point.
- Specify the **Scale** of the block by using one of the following methods:
 - Click the **Specify On-screen** checkbox. By doing so, you will specify the scale using the mouse.
 - Type the scale factor in all three directions of the insertion point. Because you can type in the scale factor, you can set the X-scale factor not equal to the Y-scale factor.
 - Another way to specify the scale is to click the **Uniform Scale** checkbox, which will allow you to input only one scale (the others will follow).

- Specify the **Rotation** of the block by using one of the following methods:
 - Click the **Specify On-screen** checkbox, which means you will specify the rotation angle using the mouse.
 - Type the rotation angle.
 - The **Block unit** part will be read-only, thereby showing you the unit that you specified when you created this block. Also, it will show the **Factor** that is based on the **Block unit** and the **drawing unit** (which is defined in the **Drawing/Units** dialog box in the **Application Menu**). With this factor, AutoCAD will automatically scale the block to suit the current drawing.
 - Click **OK** to end the command.
-  **NOTE** ■ Using the **Scale** of the block you can use negative values to insert mirror images of your block. See the following example:



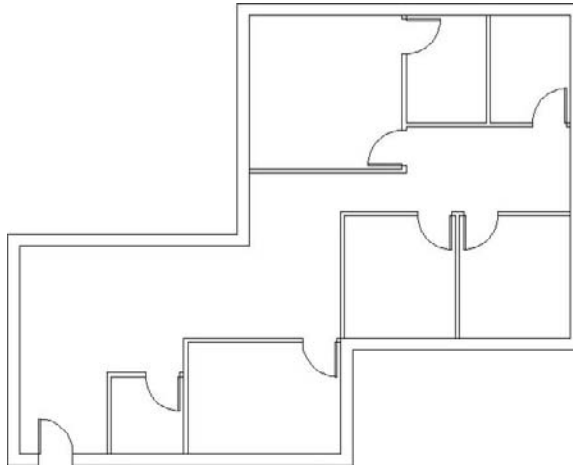
INSERTING BLOCKS (METRIC AND IMPERIAL)



Workshops 4-A and 4-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_04.dwg*.
3. Make the layer **Doors** current.

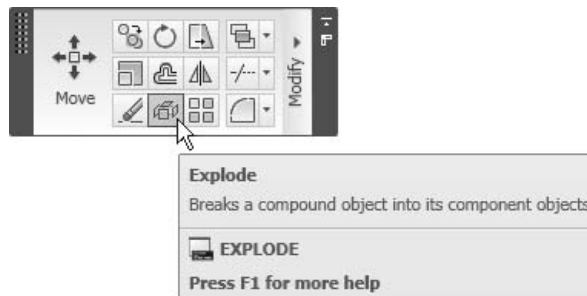
4. Using the **Insert** command, insert the block **Door** in the proper places as shown:



5. Save the file and close it.

6.4 EXPLODING BLOCKS

- When you insert incidences of blocks, keep them as blocks and do not try to change their nature.
- However, in some (rare) cases, you may want to explode the block (which is one object) to the objects forming it.
- To explode the block, you need to use the **Explode** command.
- Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Modify** panel, select the **Explode** button.



- AutoCAD will prompt you to:

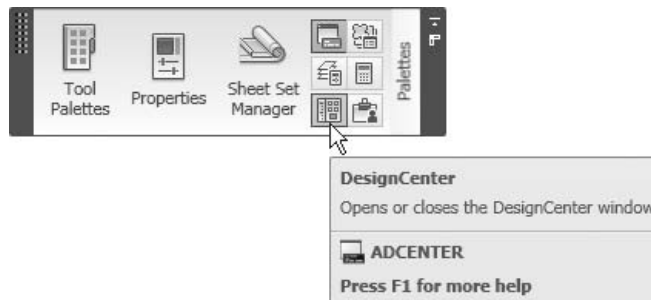
Select objects:

- Once you are done, press [Enter] or right-click.
- Once you explode a block, it will go back to its original layer (the layer the block was created in).
- Do not use this command unless you really need it.
- In older versions of AutoCAD, a block could not be defined as a cutting edge or a boundary edge in the **Trim** and **Extend** commands. Starting with AutoCAD 2005, you can select blocks as cutting edges and boundary edges. Therefore, there is no need to explode the block for this purpose.
- You can also use this command to explode polylines to lines and arcs.

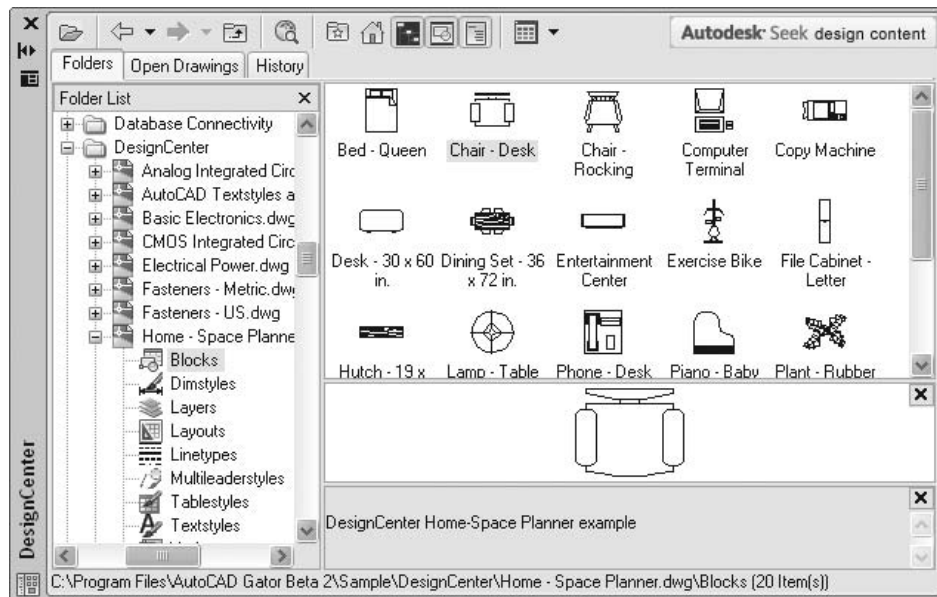
NOTE

6.5 USING DESIGN CENTER

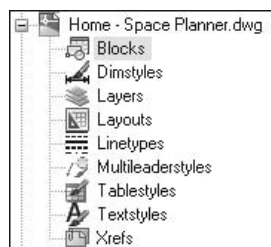
- AutoCAD has a very helpful tool called **Design Center**, which allows the user to share blocks, layers, and other things between different files.
- In this chapter, we will concentrate on blocks.
- The file you want to take the blocks from could be anywhere:
 - It could be on your computer.
 - It could be on your colleague's computer, which is hooked up to the company's local area network.
 - It could be on a website.
- To start the **Design Center** command, make sure you are in the **View** tab on the **Ribbon**, and, using the **Palettes** panel, select the **Design Center** button.



- The following will appear on the screen:



- As you can see, the **Design Center** palette is split into two parts:
 - On the left, you will see the hierarchy of your computer, including all your hard disks and network places (just like My Computer in Windows).
 - Select (by double-clicking) the desired hard disk, folder, and drawing, and you will see something like the following:



- As you can see, you can take from this drawing the following:
 - **Blocks**
 - **Dimstyles**
 - **Layers**
 - **Layouts**
 - **Linetypes**
 - **Multileaderstyles**

- **Tablestyles**
- **Textstyles**
- **Xrefs**
- Once you click the word **Blocks**, take a look at the right part of **Design Center**. You will see the blocks available in this drawing.
- There are several ways to move the blocks from this drawing to your drawing. They are:
 - Drag and drop (using the left button)
 - Drag and drop (using the right button)
 - Double-click
 - Right-click

Drag and Drop Using the Left Button

- Perform the following steps:
 - Make sure that you are in the right layer.
 - Make sure that you switched on the right **OSNAP** settings.
 - Click and hold the desired block.
 - Drag the block into your drawing; you will be holding it from the base point.
 - Once you catch the right **OSNAP**, release the mouse button.

Drag and Drop Using the Right Button

- Perform the following steps:
 - Make sure that you are in the right layer.
 - Make sure that you switched on the right **OSNAP** settings.
 - Right-click and hold the desired block.
 - Drag the block into your drawing.
 - Release the mouse button and the following shortcut menu will appear:



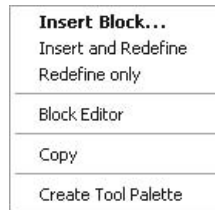
- It is the same as the **Insert** command discussed earlier.
- Follow the same steps as the **Insert** command.

Double-Click

- If you double-click any block in the **Design Center**, the **Insert** dialog box will appear and you can perform the same steps.

Right-Click

- Select the desired block and right-click. The following shortcut menu will appear:



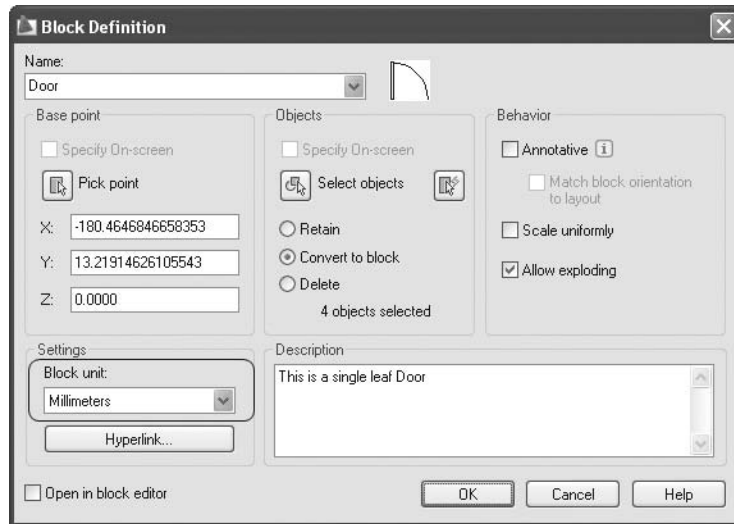
- If you select **Insert Block**, the **Insert** dialog box will appear as discussed previously.
- You will notice two options, **Insert and Redefine** and **Redefine only**. We will be discussing redefining shortly.
- Select the **Block Editor** option if you want to open this block in the block editor in order to add dynamic features to it.
- If you select **Copy**, then you will copy the block to the clipboard of Windows, and you can use it in AutoCAD or in other software. To use it, select **Edit/Paste** or [Ctrl] + V.
- We will discuss the last option in the shortcut menu, the Tool Palette, shortly.

6.6 AUTOMATIC SCALING

- If you are using the **Design Center** to bring some blocks from other drawings and find that a block is either too large or too small, you will know that there is something wrong with the **Automatic Scaling** feature.
- To control the **Automatic Scaling** feature:
 - While you are creating the block make sure you are setting the right **Block unit**.
 - Before you bring the block from the **Design Center**, set the **Units to scale inserted contents** in the **Drawing/Units** option of the **Application Menu**.

Block Unit

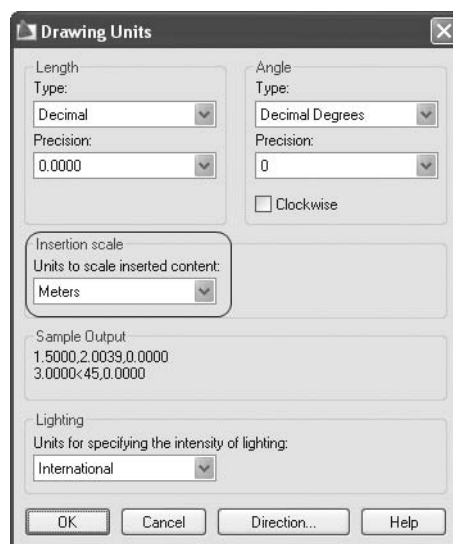
- When you are creating a block, the following dialog box will appear:



- Under the area labeled **Block unit**, select the desired unit.

Units to Scale Inserted Contents

- Before using any block, select **Drawing/Units** from the **Application Menu** and the following dialog box will appear:



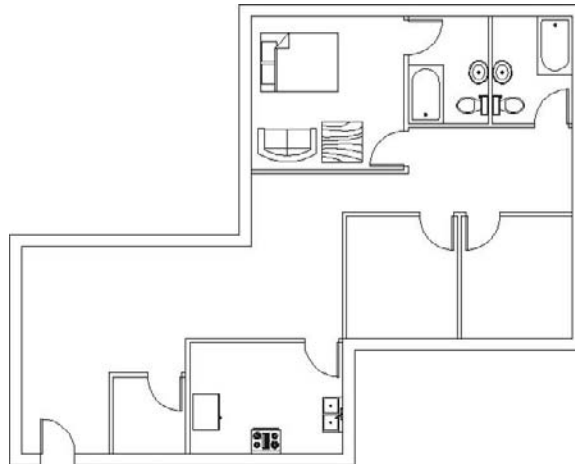
- Under the area labeled **Units to scale inserted contents**, set the desired scale used in your drawing.
- Using the two scales, AutoCAD will calculate the proper scale for the block.

USING THE DESIGN CENTER (METRIC)



Workshop 5-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_05.dwg*.
3. Make the **Furniture** layer current.
4. Select **Format/Units** and make sure that **Units to scale inserted content** is **millimeters**.
5. Open the **Design Center**. From the left part of the **Design Center** palette double-click the drive containing the AutoCAD 2010 folder.
6. Select **AutoCAD 2010/Sample/Design Center**.
7. Using *Home Space Planner.dwg*, *House Designer.dwg*, and *Kitchens.dwg* while **OSNAP** is off, drag and drop the following blocks as shown:



8. Save the file and close it.

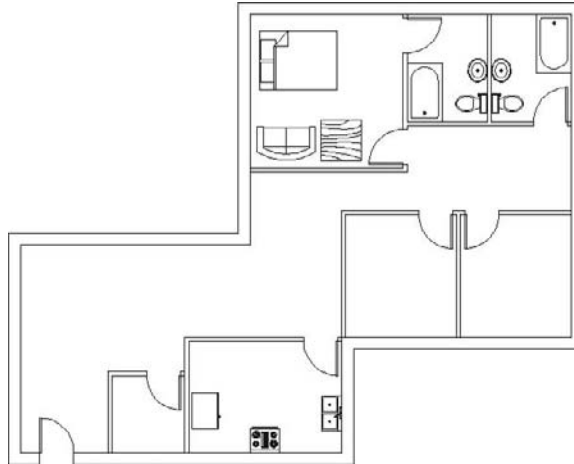
USING THE DESIGN CENTER (IMPERIAL)



Workshop 5-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_05.dwg*.
3. Make the **Furniture** layer current.
4. Select **Format/Units** and make sure that **Units to scale inserted content** is in **inches**.

5. Open the **Design Center**. From the left part of the **Design Center** palette double-click the drive containing the AutoCAD 2010 folder.
6. Select **AutoCAD 2010/Sample/Design Center**.
7. Using *Home Space Planner.dwg*, *House Designer.dwg*, and *Kitchens.dwg* while **OSNAP** is off, drag and drop the following blocks as shown:

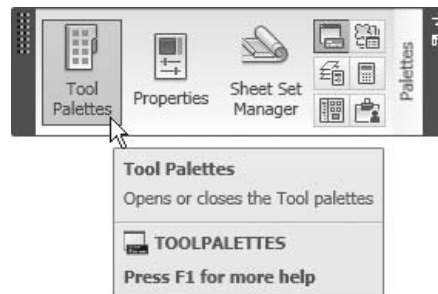


8. Save the file and close it.

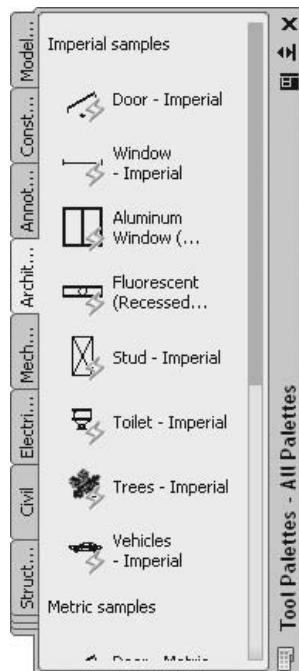
6.7 WHAT IS A TOOL PALETTE?

- The **Design Center** gives us the ability to share data from other files. However, you need to make sure you are at the right layer and specify the rotation angle, scale factor, etc. Also, you need to search for the desired content each and every time.
- **Tool Palette** will solve all of these problems.
- **Tool Palette** will keep blocks, hatch, and other items available for you regardless of which drawing you are using. You can keep virtually anything inside a Tool Palette.
- **Tool Palette** works with the same drag-and-drop method we used with the **Design Center**, but in **Tool Palette** the method will be done in two ways: from **Tool Palette** and to **Tool Palette**.
- **Tool Palette** is unique per computer in the sense that it is not limited to one particular drawing. Therefore, if you create/customize a Tool Palette, it will be available for all your drawings.

- To start the **Tool Palette** command, make sure you are in the **View** tab on the **Ribbon**, and, using the **Palettes** panel, select the **Tool Palettes** button:



- The following will appear on the screen:



- You will see several premade Tool Palettes by Autodesk® for your immediate use.
- Depending on the source, you can create your own Tool Palette using different methods.
- You can copy, cut, and paste tools inside each Tool Palette.
- You can customize the tools inside each Tool Palette.

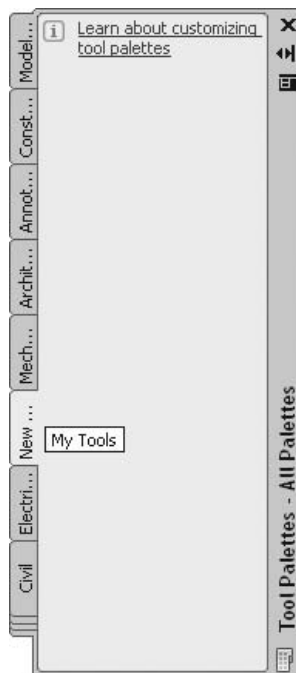
6.8 CREATING A TOOL PALETTE

Creating a Tool Palette from Scratch

- Right-click over the name of any existing Tool Palette and the following shortcut menu will appear:



- Select the **New Palette** option and a new Tool Palette will be added. Type in the name of the new Tool Palette.



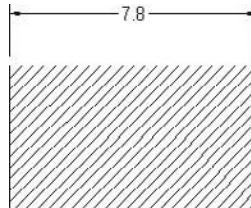
- A new empty Tool Palette will be added.
- To fill this Tool Palette, use the drag-and-drop method either from the **Graphical Area** or from the **Design Center**.



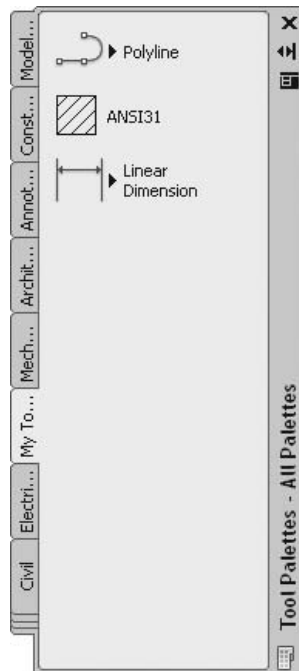
- By default, the local blocks of the current drawing are not automatically available in your Tool Palettes.

Example

- Assume we have the following drawing in front of us. This drawing contains polylines, hatch, and a linear dimension:

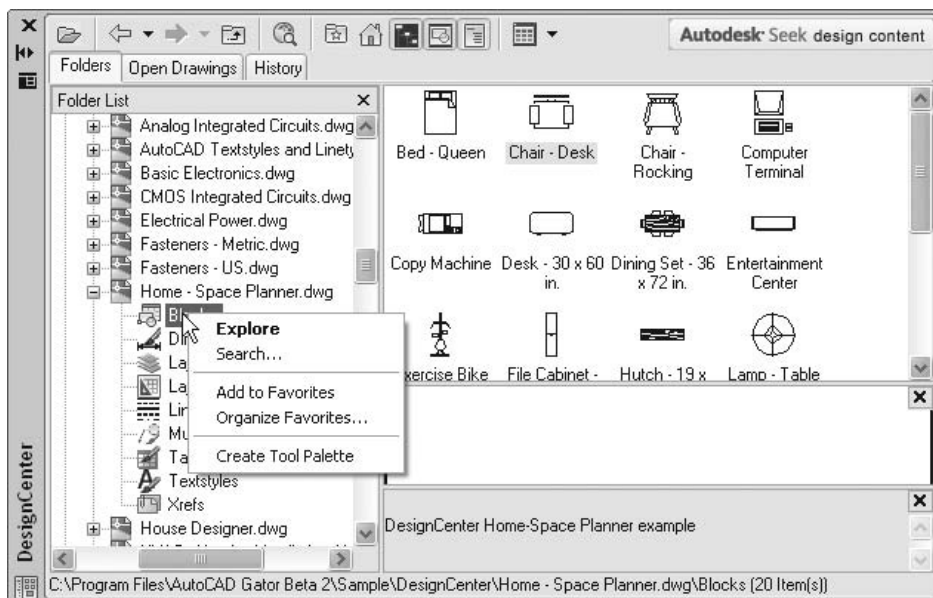


- Without issuing any commands, click on the polyline, then hold it (avoiding the blue rectangles) and drag it into the empty Tool Palette. Do the same thing for the hatch and the linear dimension. Your Tool Palette will look like the following:



Creating a Tool Palette Using the Design Center

- You can copy all of the blocks in one drawing using the **Design Center** and create a Tool Palette from them, retaining the name of the drawing.
- To do that, follow these steps:
 - Start **Design Center**.
 - Go to the desired folder, then to the desired file.
 - Right-click on the **Blocks** icon and the following shortcut menu will appear:



- Select the **Create Tool Palette** option and a new Tool Palette will be added with the name of the file you chose. It will include all of the blocks.
- You can also drag and drop any block from the **Design Center** to any desired Tool Palette.

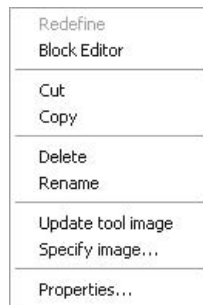
6.9 CUSTOMIZING A TOOL PALETTE

- Blocks and hatch in a Tool Palette can be copied and then pasted in the same Tool Palette or in any other Tool Palette.
- The purpose of this option is to assign different properties to each tool.

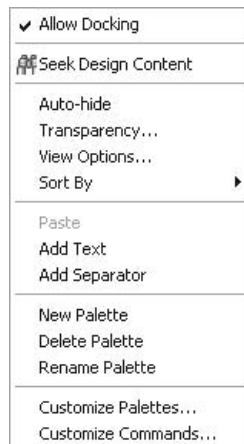
- For example, there is a block called **chair** and you want to make three copies of it. Each copy would hold a different rotation angle. The same thing applies to hatch patterns, as each copy can have a different scale factor.
- Also, you can specify that a certain block (or hatch) go to a certain layer regardless of the current layer.

How to Copy and Paste a Tool

- Follow these steps:
 - Right-click on the desired tool and the following shortcut menu will appear:

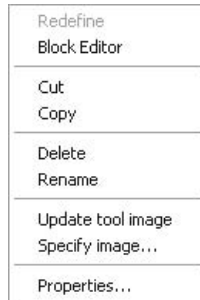


- Select the **Copy** option.
- Select the **Paste** option and the copied tool will reside at the bottom of the Tool Palette, and will hold the same name.
- In any Tool Palette, right-click and the following shortcut menu will appear:

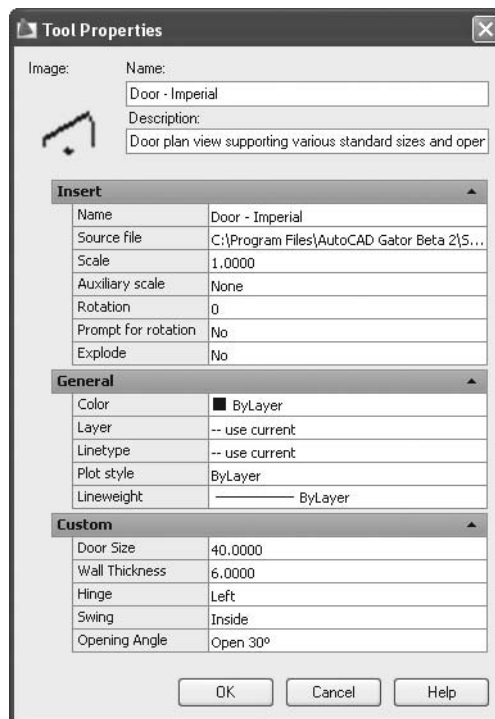


How to Customize a Tool

- Follow these steps:
 - Right-click on the copied tool and the following shortcut menu will appear:



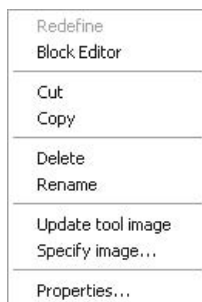
- Select the **Properties** option and the following dialog box will appear:



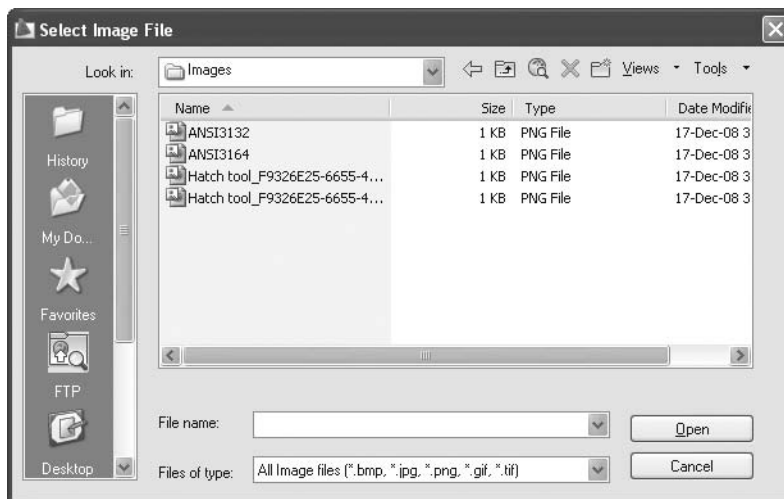
- You can change the name and description of the tool.
- There are two types of properties:
 - **Insert** properties, where a block's properties are different from a hatch's properties.



- **General** properties, such as color, layer, linetype, plot style, and lineweight.
- By default, the **General** properties are all **use current**, which means use the current settings.
- All **Insert** properties and **General** properties are changeable.
- In the **Tool Properties** dialog box, if the image of the block is not clear, you can change it. To do so, perform the following steps:
 - Go to the image, right-click, and a shortcut menu will appear:



- Select **Specify image** and the following dialog box will appear:



- Select the desired folder and the desired file and click **Open**. Now the new image will appear.

USING AND CUSTOMIZING TOOL PALETTES (METRIC AND IMPERIAL)



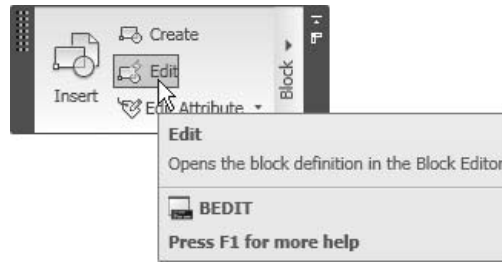
Workshops 6-A and 6-B

1. Start AutoCAD 2010.
2. Start a new file.
3. Make layer 0 current.
4. Open the **Design Center**. From the left part of the **Design Center** palette double-click the drive containing the AutoCAD 2010 folder.
5. Select **AutoCAD 2010/Sample/Design Center**.
6. Select the file *Home Space Planner.dwg*.
7. Right-click the **Blocks** icon and select **Create Tool Palette**. A new Tool Palette with the name **Home Space Planner** will be added.
8. Select the file *House Designer.dwg*.
9. Show the blocks for this file.
10. Locate **Bath Tub –26 × 60 in** block and drag and drop it in your newly created Tool Palette.
11. Do the same with the **Sink Oval Top** block.
12. Right-click on the name of the Tool Palette and select **Rename**, and change the name to **My Tools**.
13. In the **My Tools** Tool Palette, right-click on the tool named **Chair–Rocking**, and select **Properties**. In the dialog box, change the **Layer** from **use current** to **Furniture**.
14. Right-click on the same tool again and select **Copy**. Select an empty space and **Paste** it three times.
15. Using the same method in Step 13, change the rotation angle of these blocks to be 90, –90, 180.
16. Now you have a tool palette that you can use in all your drawings on this computer.

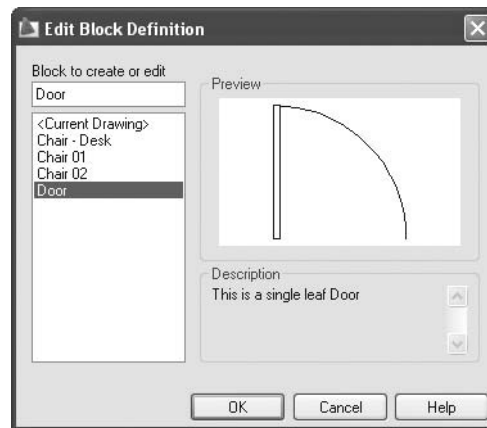
6.10 EDITING BLOCKS

- Assume that after you created a block and have inserted it into your drawing several times, you discover that there is something wrong with it.
- To solve your problem, you need to redefine the original block using **Block Editor**.
- In the block editor, you can create a **Dynamic Block** (an advanced feature in AutoCAD, but for now we will use it only to edit blocks).

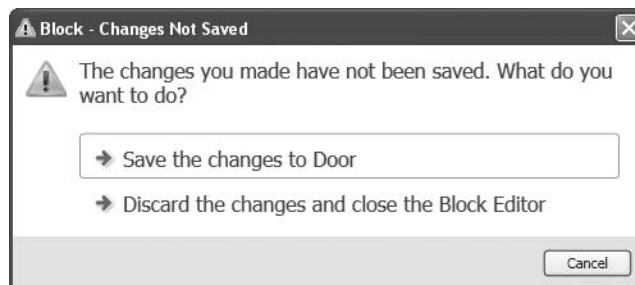
- To start the **Block Editor** command, make sure you are in the **Home** tab on the **Ribbon**, and, using the **Block** panel, select the **Edit** button.



- AutoCAD will show the following dialog box:



- Select the name of the block to be edited and click **OK**.
- The **Block Editor** will open. A new tab with many new panels will be added titled **Block Editor**.
- You should not be intimidated by these changes, simply ignore them, and make all the changes you need to your block (adding, modifying, and erasing).
- Once you are done, click the **Close Block Editor** button on the right and the following message will appear:



- You have two choices, either to save the changes to the block, or simply to discard these changes.



- If you double-click on a block, it will start the same command.

EDITING BLOCKS (METRIC)



Workshop 7-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_07.dwg*.
3. Start the **Edit** command in the **Block** panel.
4. Select one of the doors you inserted. Once the dialog box appears, click **OK**.
5. Select the arc representing the swing of the door. Right-click and select **Properties**.
6. Change its layer to **Door_Swing**.
7. Change the Linetype scale = 200.
8. Close the **Properties** palette.
9. Click **Save changes**.
10. You will see now that all the door swings change to dashed linetype.
11. Save the file and close it.

EDITING BLOCKS (IMPERIAL)



Workshop 7-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_07.dwg*.
3. Start the **Edit** command in the **Block** panel.
4. Select one of the doors you inserted. Once the dialog box appears, click **OK**.
5. Select the arc representing the swing of the door. Right-click and select **Properties**.
6. Change its layer to be **Door_Swing**.
7. Change the Linetype scale = 10.
8. Close the **Properties** palette.
9. Click **Save changes**.
10. You will see now that all the door swings change to dashed linetype.
11. Save the file and close it.

CHAPTER REVIEW

1. You should draw your original shape, which will be a block, in layer 0.
 - a. True
 - b. False
2. The **Automatic Scaling** feature in AutoCAD:
 - a. Will change the scale of the block to fit in the current drawing.
 - b. Will require two scales to convert the block.
 - c. Will work for both blocks and hatch.
 - d. None of the above.
3. Which of the following statements is true about Tool Palettes?
 - a. They can be created from blocks coming from the **Design Center**.
 - b. You can drag and drop to and from a Tool Palette.
 - c. You can customize the block inside a Tool Palette.
 - d. All of the above.
4. Which of the following commands cannot be used for blocks?
 - a. **Explode**
 - b. **Insert**
 - c. **Makelocalblock**
 - d. **Block Editor**
5. In order to make the **Design Center** and **Tool Palettes** occupy less space in the **Graphical Area**, use _____.

CHAPTER REVIEW ANSWERS

1. a
2. a
3. d
4. c
5. Auto-hide

Chapter 7 HATCHING

In This Chapter

- ◇ Hatching in AutoCAD®
- ◇ Selecting the Hatch Pattern
- ◇ Selecting the Area to be Hatched
- ◇ Previewing the Hatch
- ◇ Hatching Options
- ◇ Hatch Origin
- ◇ Advanced Features
- ◇ Hatching Using Tool Palettes
- ◇ The **Gradient** Command
- ◇ Editing an Existing Hatch or Gradient

7.1 HATCHING IN AUTOCAD®

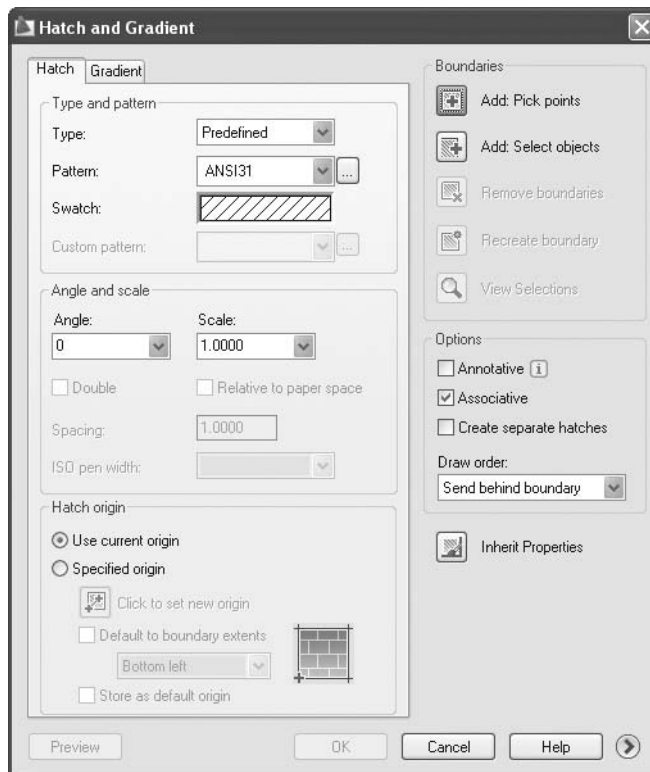
- In order to hatch in AutoCAD® you need to draw objects forming a closed area. Beginning with AutoCAD 2005, it became acceptable to hatch an area with a small opening.
- AutoCAD comes with several generic predefined hatch patterns saved in one of two files called *acad.pat* and *acadiso.pat*. You can also buy other hatch patterns from third parties, which can be found on the Internet.
- A hatch, like any other object, should be placed in a separate layer.
- There are two methods to hatch in AutoCAD: the old method (**Hatch** command) and the new method (**Tool Palette**).

7.2 SELECTING THE HATCH PATTERN

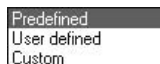
- This is the old method of hatching in AutoCAD.
- To start the **Hatch** command, make sure you are in the **Home** tab on the **Ribbon**, and, using the **Draw** panel, select the **Hatch** button.



- The following dialog box will appear:

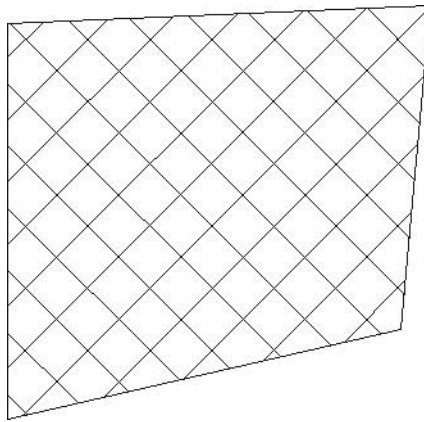


- Under the **Hatch** tab, select the **Type** pop-up list and you will see the following choices:

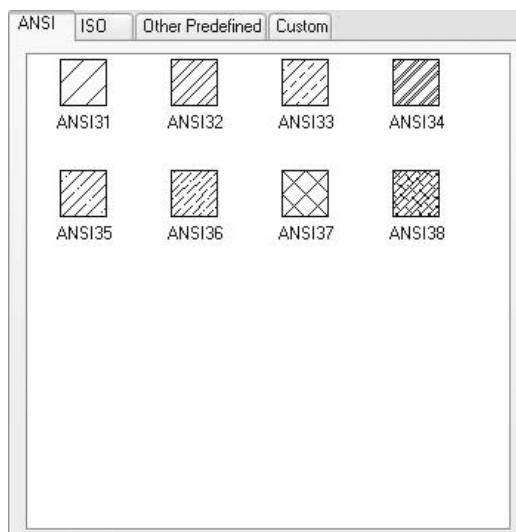


User Defined

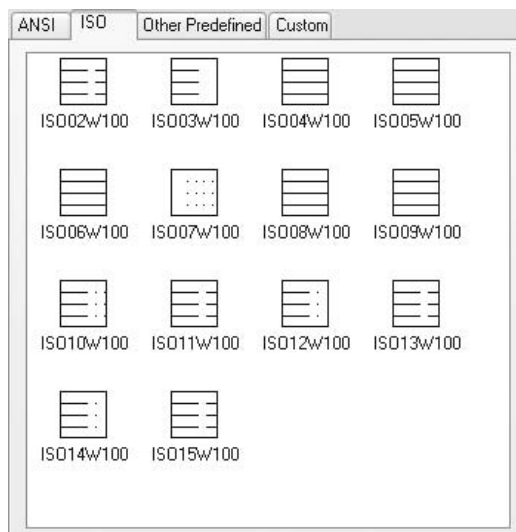
- The simplest hatch pattern has parallel lines. Once this option is selected, the following parameters will be valid:
 - **Swatch:** displays a preview of the hatch.
 - **Angle:** defines the angle of the parallel lines.
 - **Spacing:** defines the distance between two parallel lines.
 - **Double:** means “in both ways.”
- The following is a **User defined** hatch, using **Angle** = 45 and **Double** hatch:

**Predefined**

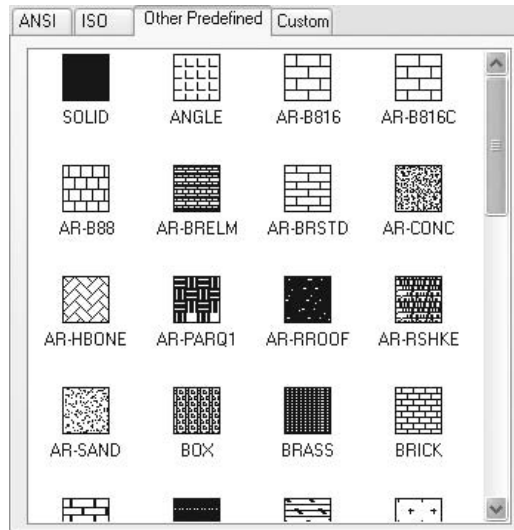
- There are a handful of hatch patterns available for use. You will see **ANSI** hatches, **ISO** hatches, and commonly used hatches. The following are the parameters to be set:
 - **Pattern:** whether you select the desired pattern using the pop-up list, select the small button with the three dots, or click **Swatch**, the following dialog box will appear showing the **ANSI** hatches:



- Click the **ISO** tab and you will see the following:



- Click the **Other Predefined** tab and you will see the following:



- When you are done selecting the hatch pattern, click **OK**.
- Set the **Angle**.
- Set the **Scale** of the hatch pattern.
- If you select one of the **ISO** hatches, set the **ISO pen width**.

Custom

- If you customized a hatch pattern or bought some from a third party, they should be used using the **Custom** tab.

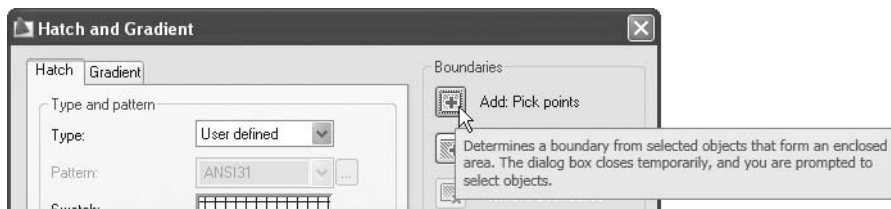
7.3 SELECTING THE AREA TO BE HATCHED

- After you pick your hatch pattern and set the relative parameters, you need to select the desired area to be hatched.
- You can select more than one area with the same command, but the hatch will be considered either as one object or as separate objects.
- There are two ways to select the objects forming the area:
 - **Add Pick points**
 - **Add Select objects**

Add Pick Points

- This method is very simple. Click inside the desired area and AutoCAD will recognize the area automatically.

- This method will also detect any objects (called islands) within the outer area, and deselect them automatically. The islands will not be hatched.
- Islands can be any object type: circle, closed polyline, text, etc.
- We will discuss island detection in detail, shortly.
- Click the **Add: Pick points** button located at the top right-hand part of the dialog box.



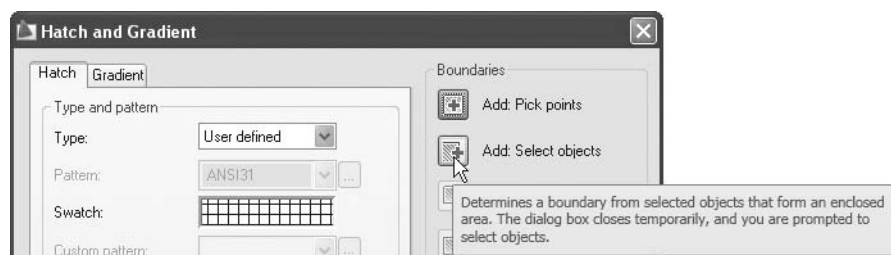
- The dialog box will disappear temporarily and the following prompt will appear:

Pick internal point or [Select objects/remove Boundaries]:

- Click inside the desired area(s). When you are done, press [Enter] or right-click and then select the **Enter** option. The dialog box will reappear.

Add Select Objects

- This is the same method used to select any object, as previously discussed.
- Click the **Add: Select objects** button located at the top right-hand part of the dialog box.

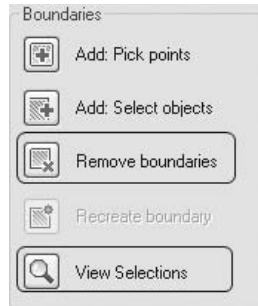


- The dialog box will disappear temporarily and the following prompt will appear:

Select objects or [pick internal point/remove Boundaries]:

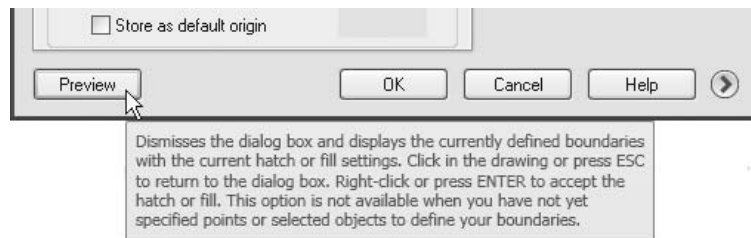
- Select the desired objects that form a closed area. When you are done, press [Enter] or right-click and then select the **Enter** option. The dialog box will reappear.

- After you have made your selection, you will be able to do two things:
 - **Remove boundaries:** you will most likely need this button when using **Add: Pick points**. **Add: Pick points** will select all the inner objects and text as islands and this button will allow you to remove some of the selected objects as islands.
 - **View Selections:** use this button to view the selection set to make sure that it is the right one.



7.4 PREVIEWING THE HATCH

- The data input is now complete, you have selected the hatch pattern along with its parameters, and you have selected the area to be hatched. The next step is to preview the hatch before you make the final decision to accept the outcome.
- Click the **Preview** button located at the lower left-hand corner of the dialog box:



- The dialog box will disappear temporarily.
- You will see the results of your settings (i.e., hatch pattern, angle, scale, islands, etc.). Accordingly, AutoCAD will prompt you to:

Pick or press Esc to return to dialog or <Right-click to accept hatch>:

- If you like what you see, right-click or press [Enter], otherwise, press [Esc].

- If you pressed [Esc], the dialog box will reappear and you can change any of the parameters, preview again, and so on.

HATCHING USING THE HATCH COMMAND (METRIC)



Workshop 8-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_08.dwg*.
3. Make the **Hatch** layer current.
4. Start the **Hatch** command and select the **Type** as **Predefined**.
5. Click on **Swatch** and select the **Other Predefined** tab. Select the **AR-CONC** pattern.
6. Set the **Scale** = 100.
7. Click on the **Add: Pick points** button, and click inside the area representing the outer wall. Press [Enter].
8. Click **Preview** to see the results of the hatching, then press [Enter] to end the command.
9. Start the **Hatch** command again and select the **Type** as **Predefined**.
10. Click on **Swatch** and select the **ANSI** tab. Select the **ANSI32** pattern.
11. Set the **Scale** = 500.
12. Click on the **Add: Pick points** button, and click inside the area representing the inner walls. Press [Enter].
13. Click **Preview** to see the results of the hatching, then press [Enter] to end the command.
14. Save the file and close it.

HATCHING USING THE HATCH COMMAND (IMPERIAL)



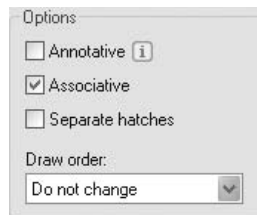
Workshop 8-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_08.dwg*.
3. Make the **Hatch** layer current.
4. Start the **Hatch** command and select the **Type** as **Predefined**.
5. Click on **Swatch** and select the **Other Predefined** tab. Select the **AR-CONC** pattern.
6. Set the **Scale** = 5.

7. Click on the **Add: Pick points** button and click inside the area representing the outer wall. Press [Enter].
8. Click **Preview** to see the results of the hatching, then press [Enter] to end the command.
9. Start the **Hatch** command again and select the **Type** as **Predefined**.
10. Click on **Swatch** and select the **ANSI** tab. Select the **ANSI32** pattern.
11. Set the **Scale** = 20.
12. Click on the **Add: Pick points** button and click inside the area representing the inner walls. Press [Enter].
13. Click **Preview** to see the results of the hatching, then press [Enter] to end the command.
14. Save the file and close it.

7.5 HATCHING OPTIONS

- While you are hatching, there are some options in the **Hatch** command that you should know in order to have full control over the hatching process.
- At the right side of the dialog box, there is a part labeled **Options**:



Annotative

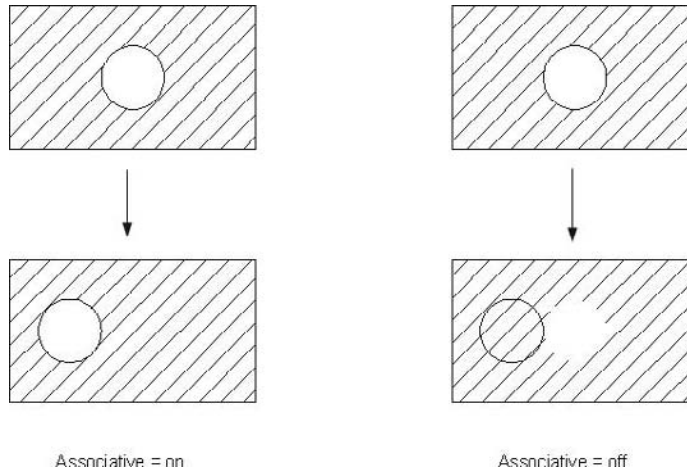
- This is an advanced feature related to printing.

Associative

- Associative means that there is a relationship between the hatch and the boundary. Whenever the boundary changes, the hatch changes automatically. It is recommended to keep this option selected.

Example

- Assume you have the following shape, which you want to hatch:

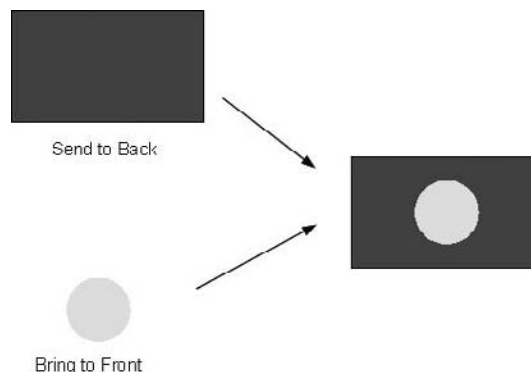


Create Separate Hatches

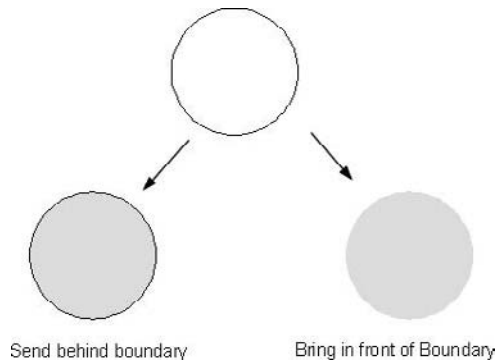
- By default, when you hatch several areas using the same command, all of the hatches will be considered one object. They will be moved together and erased together.
- Using the **Create separate hatches** option, you can hatch several areas using the same command and each hatch will be considered a separate object. To do so, click the **Create separate hatches** checkbox.

Draw Order

- If hatched areas are intersecting with other hatched areas (specifically hatched with **Solid** hatching), you need to set the **Draw** order while you are inserting the hatch to ensure the right appearance of each area.
- The four options are:
 - **Send to back** (see the following)
 - **Bring to front** (see the following)

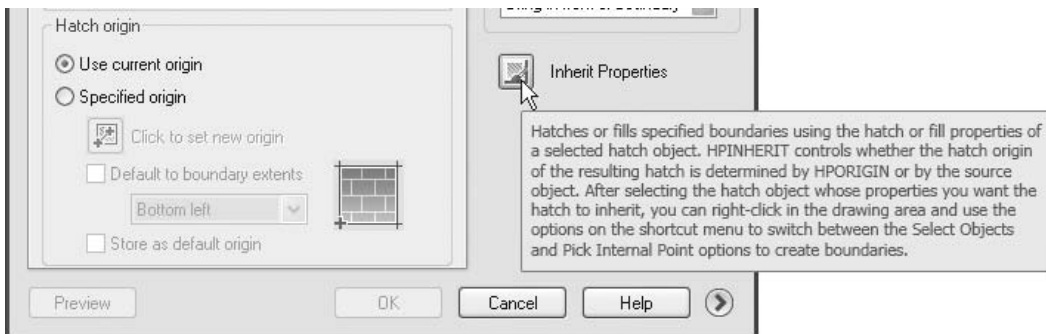


- **Send behind boundary** (see the following)
- **Bring in front of boundary** (see the following)



Inherit Properties

- Below the **Options** part is the **Inherit Properties** button:



- The purpose of this button is to help users quickly hatch a new area with the same exact features of an existing hatch.
- Click the **Inherit Properties** button and the dialog box will disappear temporarily. The following prompt will appear:

Select hatch object:

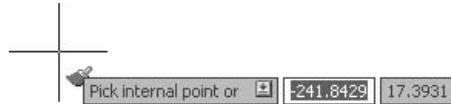
- The mouse will change to a paintbrush:



- Click the source hatch pattern and the following steps will occur.
 - A new prompt will appear:

Inherited Properties: Name <SOLID>, Scale <1.0000>,
 Angle <0>
 Pick internal point or [Select objects/remove
 Boundaries]:

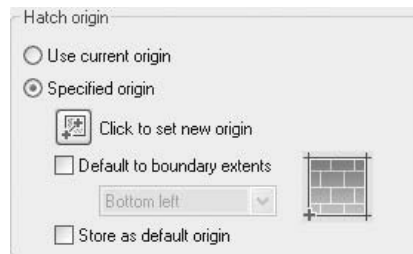
- Meanwhile, the cursor will change to a bigger paintbrush, with small crosshairs, so you can click inside the new areas you want to hatch:



- Click inside the desired area(s). When you are done, press [Enter] or right-click and select the **Enter** option. The dialog box will reappear. Click **OK** to finish the command.

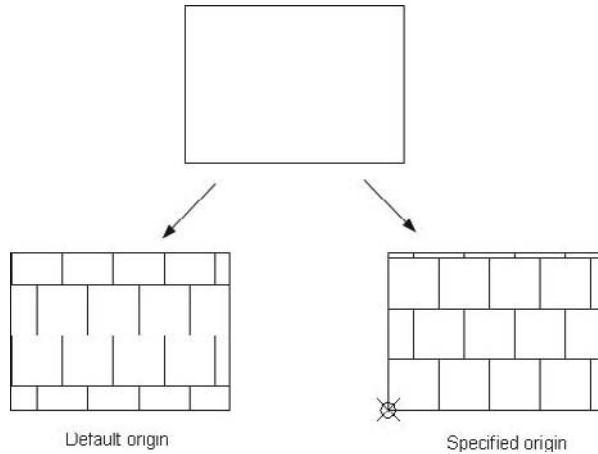
7.6 HATCH ORIGIN

- By default, AutoCAD uses 0,0 as the starting point for any hatch. Therefore, if you want to hatch using a brick-like hatch, and you need it to start from a point in the area you want to hatch, you must use the settings in this part of the dialog box.
- At the lower left-hand side of the dialog box you will see a part labeled **Hatch origin**:



- Click the **Specified origin** radio button.
- In order to specify a new origin, you have two options:
 - Use the **Click to set new origin** button and select any point you desire.
 - Click the **Default to boundary extents** checkbox and select one of the options available: Bottom left, Bottom right, Top left, Top right, or Center. You can also store this choice as the default origin.

- See the following example:



ASSOCIATIVE HATCHING AND HATCH ORIGIN (METRIC)



Workshop 9-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_09.dwg*.
3. Make the **Hatch** layer current.
4. Zoom to the kitchen at the lower right of the plan.
5. Start the **Hatch** command and select the **Type** as **Predefined**.
6. Click on **Swatch** and select the **Other Predefined** tab. Select the **ANGLE** pattern.
7. Set the **Scale** = 1000. Set the **Hatch origin** = Use current origin. Also, make sure that **Associative** is on.
8. Click on the **Add: Pick points** button and click inside the area representing the bathroom. Press [Enter].
9. Click **Preview** to see the results of the hatching. Press [Esc] and you will see the dialog box again. Select **Specified origin** and specify the lower left corner of the kitchen to be the new origin. Click **Preview** again. Do you see a change? Press [Enter] to finish the command.
10. Move the oven to the right. What is happening to the hatch? Does it react correctly or not?
11. Save the file and close it.

ASSOCIATIVE HATCHING AND HATCH ORIGIN (IMPERIAL)



Workshop 9-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_09.dwg*.
3. Make the **Hatch** layer current.
4. Zoom to the kitchen at the lower right of the plan.
5. Start the **Hatch** command and select the **Type** as **Predefined**.
6. Click on **Swatch** and select the **Other Predefined** tab. Select the **ANGLE** pattern.
7. Set the **Scale** = 50. Set the **Hatch origin** = Use current origin. Also, make sure that **Associative** is on.
8. Click on the **Add: Pick points** button and click inside the area representing the bathroom. Press [Enter].
9. Click **Preview** to see the results of the hatching. Press [Esc] and you will see the dialog box again. Select **Specified origin** and specify the lower left-hand corner of the kitchen to be the new origin. Click **Preview** again. Do you see a change? Press [Enter] to finish the command.
10. Move the oven to the right. What is happening to the hatch? Does it react correctly or not?
11. Save the file and close it.

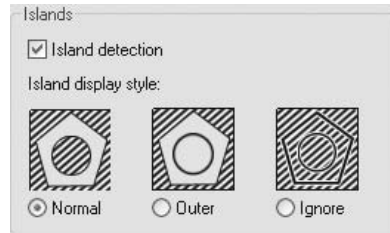
7.7 ADVANCED FEATURES

- At the lower right-hand corner of the dialog box, you will see a small button with an arrow pointing to the right. Click this button to see more advanced options for hatching:



Islands

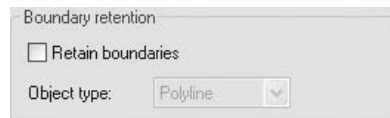
- The first is the **Islands** option:



- Islands in AutoCAD are the objects inside the outer boundary of an area to be hatched.
 - Click **Island detection** off if you do not want AutoCAD to recognize the inner objects.
 - Click **Island detection** on if you want AutoCAD to select the inner objects as boundaries and do not want to hatch them.
- Select one of three styles:
 - **Normal:** If three or more objects are nested inside each other, AutoCAD will hatch the outer, leave the second, hatch the third, etc.
 - **Outer:** If three or more objects are nested inside each other, AutoCAD will only hatch the outer, leaving the inner objects intact.
 - **Ignore:** If three or more objects are nested inside each other, AutoCAD will ignore all the inner objects and only hatch the outer area.

Boundary Retention

- By default, AutoCAD creates a polyline around the detected area. Once the **Hatch** command is finished, AutoCAD will delete this polyline.
 - The **Boundary retention** option allows you to keep this temporary polyline.



- Click the **Retain boundaries** checkbox and specify the **Object type** as either **Polyline** or **Region**.

Boundary Set

- When using the **Add: Pick points** option to define the boundary to be hatched, AutoCAD will analyze all of the objects in the **Current viewport** by default.
- This could take a long time depending on the complexity of the drawing. To minimize the time, you can provide a selection set for AutoCAD to analyze the boundary.

- The **Boundary** set part of the dialog box looks like this:



- By default, the selected option is **Current viewport**.
- Click the **New** button and the dialog box will disappear temporarily. The following prompt will appear:

Select objects:

- Select the desired objects and press [Enter] or right-click. The dialog box will appear again, but this time the selected option will be **Existing set**, as follows:



- Now, when you ask AutoCAD to select a boundary by clicking inside it, AutoCAD will not analyze all of the objects in the current viewport, but instead will only analyze the objects in the set you have selected.

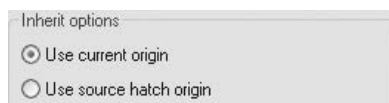
Gap Tolerance

- AutoCAD accepts an area with a small gap to hatch. The size is defined by the user.
- You can set the maximum gap that AutoCAD may ignore:



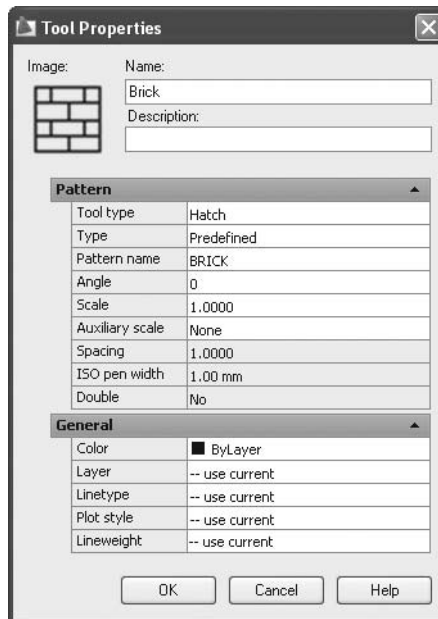
Inherit Options

- This section deals with two things previously discussed: **Inherit Properties** and **Hatch origin**.
- When you select **Inherit Properties** from an existing hatch, you have two options to choose from:
 - **Use current origin**
 - **Use source hatch origin**



7.8 HATCHING USING TOOL PALETTES

- Previously, we introduced the concept of **Tool Palettes**, which includes both blocks and hatches.
- The main feature of **Tool Palettes** is the drag-and-drop feature. We will utilize this feature to speed up the process of hatching.
- Perform the following steps:
 - Create a new Tool Palette (for example, call it “My Hatches”).
 - Use the **Hatch** command to add hatches to your different drawings.
 - While you are hatching, you are changing the hatch settings.
 - Each time you would like this hatch with all of its settings to be used in other drawings, simply drag and drop it into your newly created Tool Palette.
 - You can make several copies of your hatch. Then, you can customize the different settings of each hatch. Right-click on any hatch in your Tool Palette and select **Properties**. The following dialog box will appear:



- After several drawings, you will have a large library of hatches that you can use in your drawings.
- Now, use the drag-and-drop feature to move hatches from your Tool Palettes to your drawing.
- By default, there will be a Tool Palette called **Hatches and Fills** that you can use if you do not wish to create your own.

HATCHING AND TOOL PALETTE (METRIC AND IMPERIAL)

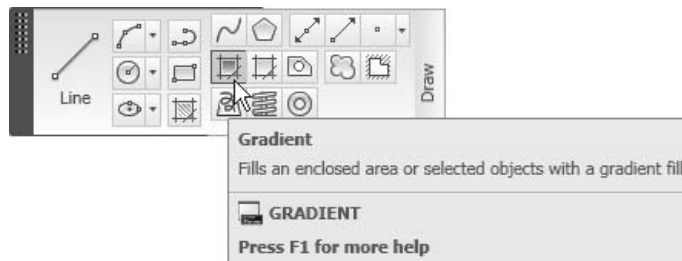


Workshop 10-A and 10-B

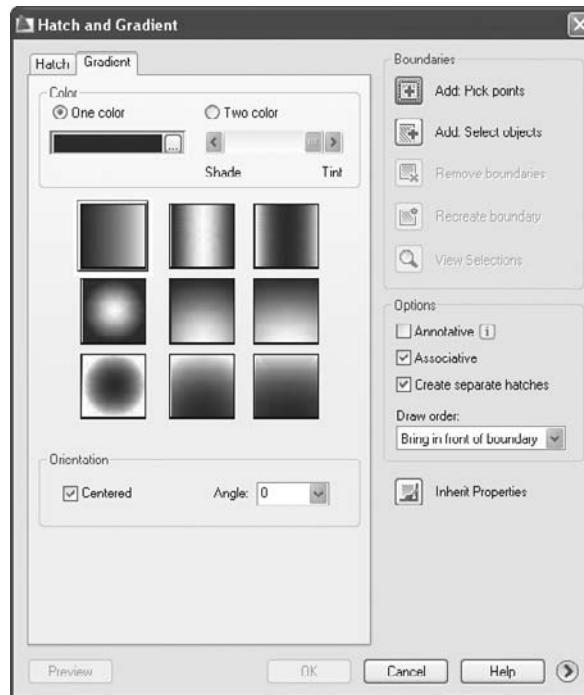
1. Start AutoCAD 2010.
2. Open the file *Workshop_10.dwg*.
3. Start **Tool Palettes**. Create a new Tool Palette and name it **My Hatches**.
4. Drag and drop the three hatches we used in our file, which are: **AR-CONC**, **ANSI32**, and **ANGLE**.
5. In the new Tool Palette, select any of the three hatches and right-click. Select **Properties** and make sure that the current layer is always **Hatch** and not **use current**.
6. Next time you use the hatch from the Tool Palette, you will not worry about which layer the hatch will reside in.
7. Save the file and close it.

7.9 THE GRADIENT COMMAND

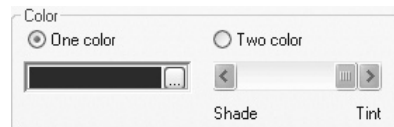
- Use the **Gradient** command if you want to shade a 2D area with one color along with white, black, a shade of gray, or with two colors.
- The **Gradient** command uses the same method as the **Hatch** command, so there is no need to describe the steps again.
- To start the **Gradient** command, make sure you are in the **Home** tab on the **Ribbon**, and, expanding the **Draw** panel, select the **Gradient** button.



- The following dialog box will appear:



- Begin by specifying whether you want to use one color along with white, or two colors:



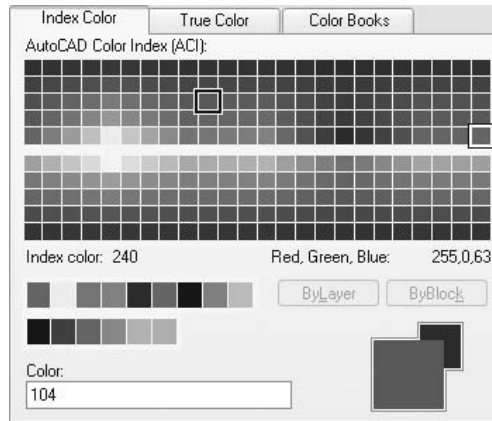
- If you select **One color**, then you will need to do the following:
 - Click the small button (with the three dots in it) to select the desired color.
 - Move the slider from **Tint** (total white) to **Shade** (total black), or any color in between the two.
- If you select **Two color**, the following will appear:



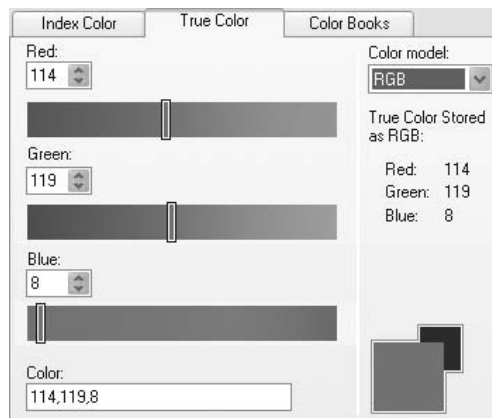
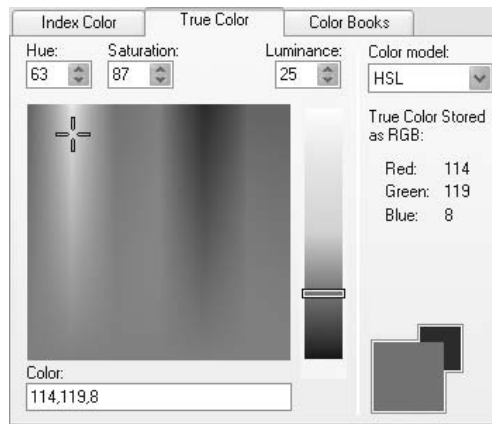
Selecting a Color

- There are three sets of colors you can select from: **Index Color**, **True Color**, and **Color Books**.

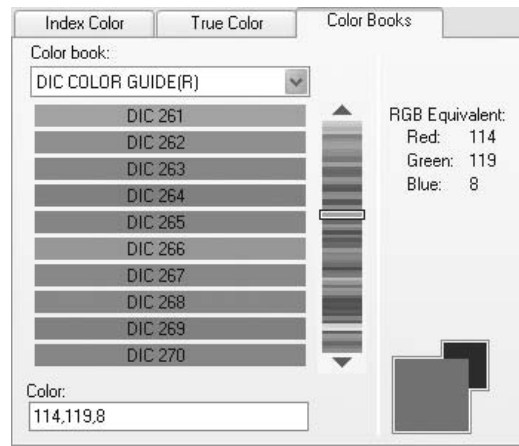
- 255 colors, called **Index Color**.



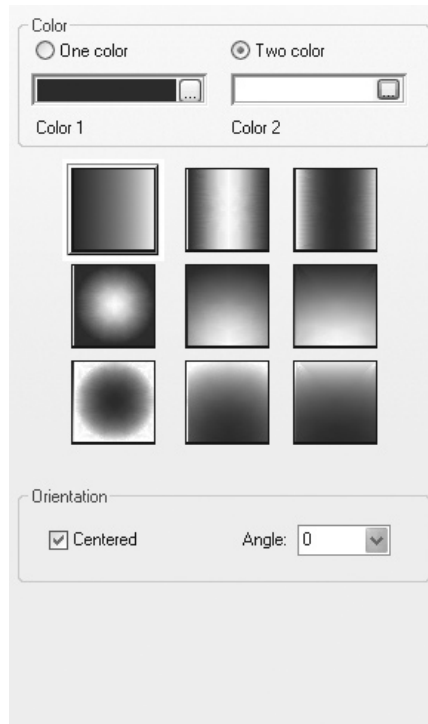
- 24-bit **True Color**, where you can select from two models: **HSL** (Hue, Saturation, and Luminance) or **RGB** (Red, Green, and Blue). See the following for examples of each.



- You can also select a color from one of 11 **Color Books** available:



- Now, select one of the nine gradient patterns:



- Select whether your pattern is symmetrical (**Centered**) or not, and then select the angle of the pattern.
- The rest of the steps are identical to the **Hatch** command.

USING THE GRADIENT COMMAND

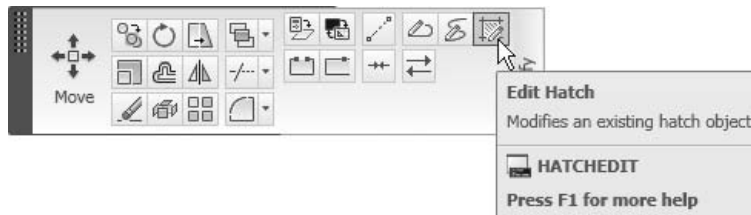


Exercise 33

1. Start AutoCAD 2010.
2. Open the file *Exercise_33.dwg*.
3. Start the **Gradient** command.
4. Select the **Two color** option.
5. Select the first color to be 142 and the second to be 61.
6. Choose the upper-middle method to define how the two colors should be mixed.
7. Select the inner area.
8. Preview the gradient.
9. Save the file and close it.

7.10 EDITING AN EXISTING HATCH OR GRADIENT

- Use the **Edit Hatch** command if you want to edit an existing hatch (whether you are using the **Hatch** command, the drag-and-drop method, or the **Gradient** command).
- To start the **Edit Hatch** command, make sure you are in the **Home** tab on the **Ribbon**, and, expanding the **Modify** panel, select the **Edit Hatch** button.



- There are other ways to reach the same command:
 - Right-click on the desired hatch. From the shortcut menu select the **Hatch Edit** option.
 - Double-click the desired hatch.
- Regardless of the method used, AutoCAD will show the following prompt:

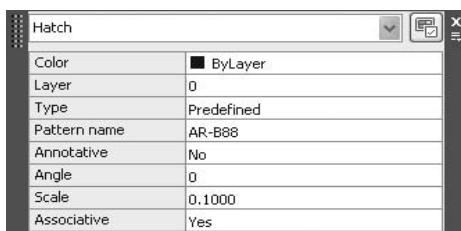
Select hatch object:

- Select the desired hatch and the **Hatch** dialog box will appear. Change the settings as you wish and click **OK**.

- Other ways to edit a hatch or gradient include:
 - **Quick Properties**
 - **Properties**
 - Double-clicking

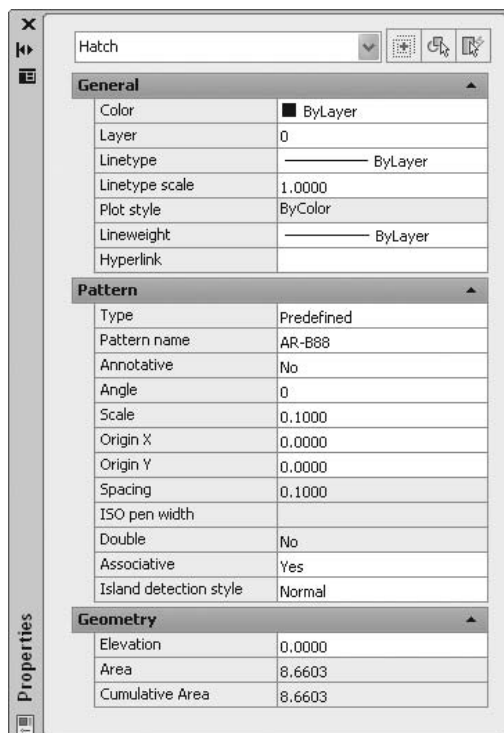
Quick Properties

- Whenever you select a hatch, the **Quick Properties** window will appear. You can make the most important edits you need here:



Properties

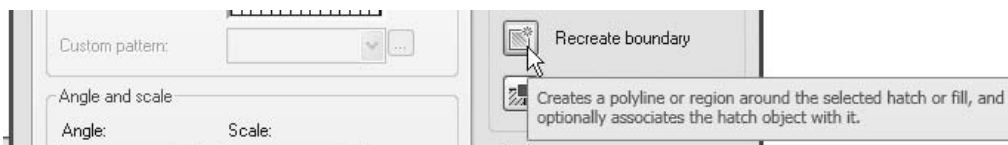
- Select the hatch, right-click, and select **Properties** from the shortcut menu. The following **Properties** palette will appear:



- In the **Properties** palette you can edit all of the data related to the hatch selected (some of the settings will be applicable to user-defined hatch patterns such as **Spacing** and **Double**).

Recreate Boundary

- If you hatched an area, and for some reason the boundary was lost but the hatch was kept, you can use this option to recreate the boundary of an existing hatch.
- Double-click the hatch without the boundary. The **Hatch** dialog box will appear. Perform the following steps:
 - Click the **Recreate boundary** button. The dialog will disappear temporarily.



- The following prompt will appear:

Enter type of boundary object [Region/Polyline]
<Polyline>:

- Type **P** for polyline or **R** for region, and the following prompt will appear:

Associate hatch with new boundary? [Yes/No] <Y>:

- Type **Y** for yes or **N** for no.
- The dialog box will appear again. Click **OK** to finish the command.

EDIT HATCHING (METRIC)



Workshop 11-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_11.dwg*.
3. Select the **AR-CONC** hatch, right-click, and then select **Hatch Edit**. The **Hatch Edit** dialog box will appear. Set the **Scale** = 75.
4. Click **Preview** and press [Enter] to accept the changes you made.
5. Double-click the **ANGLE** hatch and set the **Angle** = 45.
6. Click **Preview** and press [Enter] to accept the changes you made.
7. Save the file and close it.

EDIT HATCHING (IMPERIAL)



Workshop 11-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_11.dwg*.
3. Select the **AR-CONC** hatch, right-click, and then select **Hatch Edit**. The **Hatch Edit** dialog box will appear. Set the **Scale** = 2.5.
4. Click **Preview** and press [Enter] to accept the changes you made.
5. Double-click the **ANGLE** hatch and set the **Angle** = 45.
6. Click **Preview** and press [Enter] to accept the changes you made.
7. Save the file and close it.

CHAPTER REVIEW

1. The origin of a hatch area is defined by AutoCAD and you cannot change it.
 - a. True
 - b. False
2. You can create a boundary for an existing hatch using the _____ button.
3. AutoCAD now supports 24-bit **True Color**.
 - a. True
 - b. False
4. Which of these statements is NOT true about hatching in AutoCAD?
 - a. You can set the **Draw** order of the hatch.
 - b. You can use the conventional method and the drag-and-drop method.
 - c. The desired area must be closed in order to hatch it.
 - d. You can use hatch or gradient colors.
5. Which of the following tasks cannot be done using the **Hatch** command?
 - a. Separate hatches using the same command.
 - b. Hatching areas with gap.
 - c. Set the scale of the hatch pattern.
 - d. Hatch with a three-color gradient.
6. If you want the hatch to react to any change in the boundary, click the _____ checkbox in the **Hatch** dialog box.

CHAPTER REVIEW ANSWERS

1. b
2. Recreate boundary
3. a
4. c
5. d
6. Associative

Chapter 8 TEXT AND TABLES

In This Chapter

- ◇ Introduction
- ◇ **Text Style**
- ◇ **Single Line Text**
- ◇ **Multiline Text**
- ◇ An Introduction to Editing Text
- ◇ Editing Text Using **Quick Properties** and **Properties**
- ◇ Text and Grips
- ◇ **Check Spelling** and **Find and Replace**
- ◇ **Table Style**
- ◇ The **Table** Command

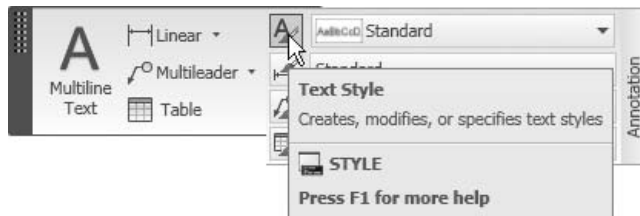
8.1 INTRODUCTION

- In order to input text in AutoCAD® you first need to create your own text style.
- In **Text Style**, you will specify the characteristics of all the text you input into your drawing file.
- You should have several text styles in your drawing in order to cover all of the requirements (large fonts for titles, small fonts for remarks, special text style for dimensions, etc.).
- Text styles can be shared between files using **Design Center**.
- After you create your text style, you can use several commands to input text into your drawing:
 - **Single line text** (old method)
 - **Multiline text** (new method)
- After you finish writing, you can edit and check the spelling of the text.
- In order to create tables with text, you have to create a **Table Style**.

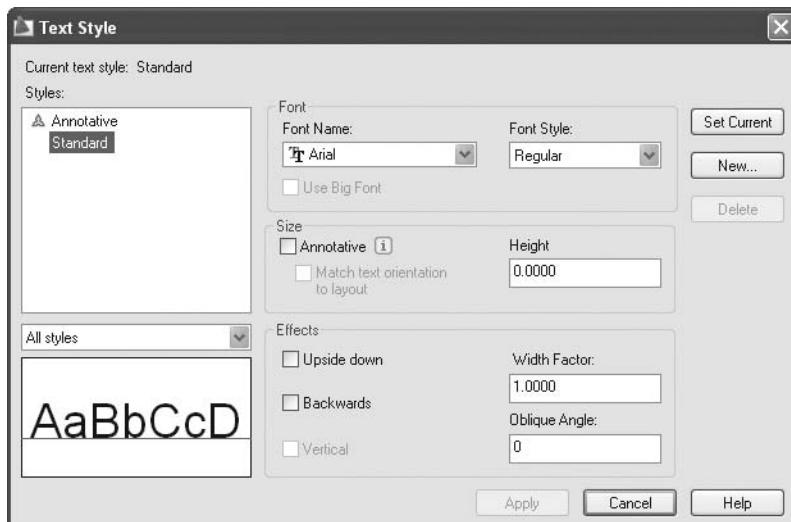
- You can insert tables and write text inside them.
- Table styles can be shared between files using **Design Center**.

8.2 TEXT STYLE

- The first step in writing text in AutoCAD is to create a text style.
- **Text Style** is where you define the characteristics of your text.
- To start the **Text Style** command, make sure you are in the **Home** tab on the **Ribbon**, and, expanding the **Annotation** panel, select the **Text Style** button:



- The following dialog box will appear:



- As you can see, AutoCAD comes with a default text style called **Standard**.
- This style contains the default settings of the text style.
- This style is very simple and you should consider creating your own.
- To create a new text style, click the **New** button and the following dialog box will appear:



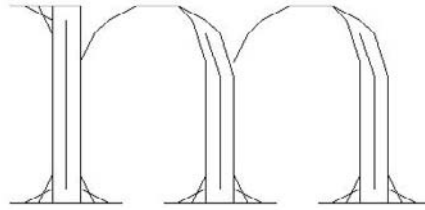
- Type in the name of the new text style using the same naming convention you used with the layers.
- When done, click **OK**.

Font Name

- The first thing to do is to select the desired font.
- There are two types of fonts you can use in AutoCAD:
 - **Shape Files** (*.shx), the old type of fonts.
 - **True Type** fonts (*.ttf), the new type of fonts.
- The following illustration shows the difference between the two types:



True Type Font



Shape File

Font Style

- If you select a **True Type** font, you will be able to select the **Font Style**. You can choose from the following:
 - Regular
 - Bold
 - Bold Italic
 - Italic
- See the following illustration:

m

Regular

m

Bold

m

Bold/Italic

m

Italic

Annotative

- This is an advanced feature; we will leave it off for this example.

Height

- Specify the **height** of the text. See the following illustration:



- As you can see, the height mentioned in the dialog box is for the capital letters.
- There are two methods for specifying the height of text:
 - Leave the value equal to 0 (zero), which means you have to specify the height each and every time you use this style.
 - Specify a height value that will always be used once you have created this style.

Effects

- There are five effects you can add to your text:
 - Upside down** (see the following illustration):



- Backwards**, which is used to write from right to left:



- Width Factor**, which affects the width/height:



- **Oblique Angle** (see the following):

Bag

Regular

Bag

Oblique Angle = +15

Bag

Oblique Angle = -15



- **Vertical** is only applicable for *.shx fonts—it will write the text from top to bottom (good for any *.shx Chinese fonts).
- When you are done, click the **Apply** button, and then the **Close** button.
- At the left part of the dialog box, there is a pop-up list showing **All styles**. Depending upon your specifications, this list will show you all of the defined text styles—whether they are used or not used—or only the text styles that are used in the current drawing.

CREATING TEXT STYLES (METRIC)



Workshop 12-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_12.dwg*.
3. Create a text style named “Title” with the following settings:
 - a. **Font** = Arial
 - b. **Font Style** = Bold
 - c. **Height** = 900
 - d. **Width Factor** = 2
4. Create a text style named “Inside_Annot” with the following settings:
 - a. **Font** = Times New Roman
 - b. **Font Style** = Regular
 - c. **Height** = 300
 - d. **Width Factor** = 1
5. Create a text style named “Dimension” with the following settings:
 - a. **Font** = Arial
 - b. **Font Style** = Regular
 - c. **Height** = 400
 - d. **Width Factor** = 1
6. Save the file and close it.

CREATING TEXT STYLES (IMPERIAL)

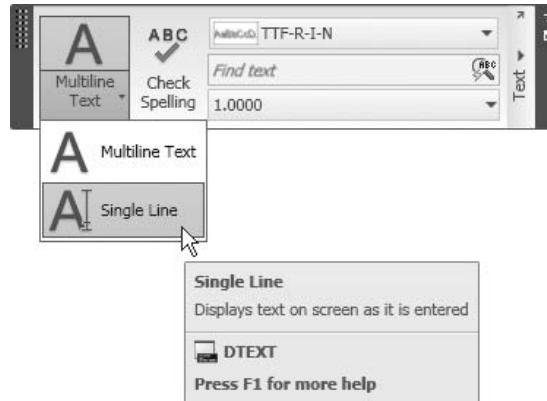


Workshop 12-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_12.dwg*.
3. Create a text style named “Title” with the following settings:
 - a. **Font** = Arial
 - b. **Font Style** = Bold
 - c. **Height** = 3'-0"
 - d. **Width Factor** = 2
4. Create a text style named “Inside_Annot” with the following settings:
 - a. **Font** = Times New Roman
 - b. **Font Style** = Regular
 - c. **Height** = 1'-0"
 - d. **Width Factor** = 1
5. Create a text style named “Dimension” with the following settings:
 - a. **Font** = Arial
 - b. **Font Style** = Regular
 - c. **Height** = 1'-4"
 - d. **Width Factor** = 1
6. Save the file and close it.

8.3 SINGLE LINE TEXT

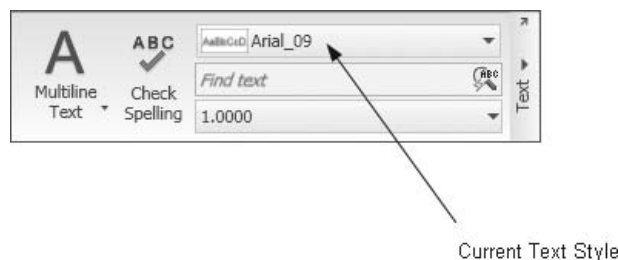
- **Single Line Text** is the first of two commands you can use to write text in AutoCAD.
- Although you write several lines of text in each command, each line would be considered a separate object.
- To start the **Single Line Text** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Text** panel, select the **Single Line** button:



- The following prompt will appear:

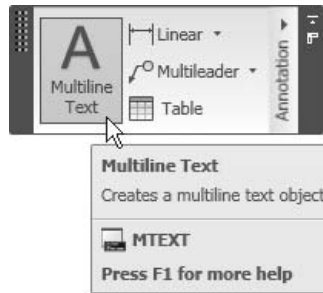
Current text style: "arial_09" Text height:
0.9000 Annotative No
Specify start point of text or [Justify/Style]:
(Specify the start point of the baseline)
Specify rotation angle of text <0>: **(Specify the rotation
angle of the baseline, then press [Enter] and start writing)**

- You will see the text on the screen. Press [Enter] once each time you want a new line; press [Enter] twice to end the command.
- As previously noted, AutoCAD will use the current text style to write the desired text. In the **Text** panel, you will see something like the following:



8.4 MULTILINE TEXT

- The **Multiline Text** command simulates Microsoft® Word simplicity in creating text. It is easier for people who have experience using Word.
- All the text you write in a single command would be considered a single object.
- To start the **Multiline Text** command, make sure you are in the **Home** tab on the **Ribbon**, and, using the **Annotation** panel, select the **Multiline Text** button.



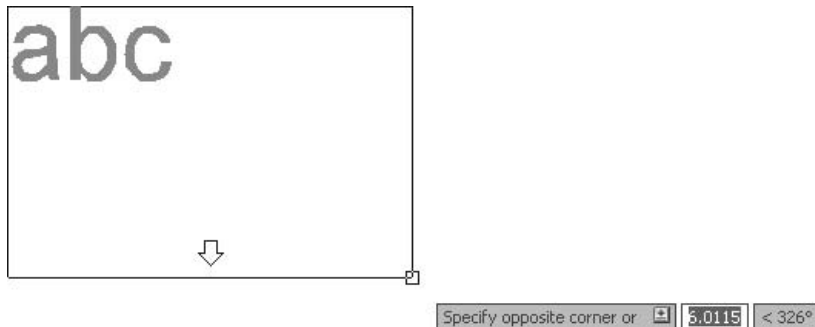
- The following prompt will appear:

Current text style: "arial_09" Text height: 0.9000
 Annotative No
 Specify first corner: **(Specify first corner)**
 Specify opposite corner or [Height/Justify/Line spacing/
 Rotation/Style/Width]: **(Specify opposite corner)**

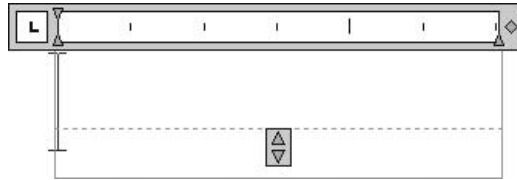
- At the first prompt, the cursor will change to a crosshair:



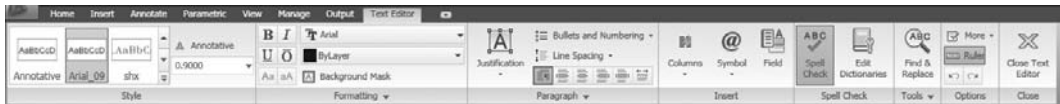
- After you specify the first point, you will get something like the following:



- AutoCAD wants you to select two opposite corners to specify the area that you will write in.
- After you specify the two corners, a text editor with a ruler will appear:



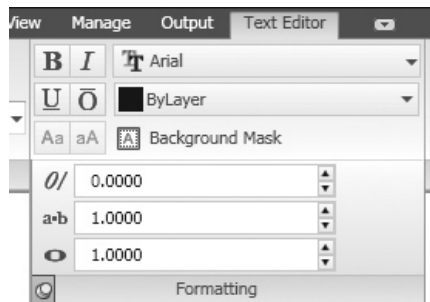
- AutoCAD will automatically present a new tab called **Text Editor** (that will disappear when you are done with this command), which looks like the following:



- A blinking cursor will appear in the text editor so you can type your desired text. Using the **Text Editor** tab, you can format the text as you wish.
- If you created a text style (highly recommended), then at the left part of the tab you will see the name of the current text style along with the height.
- In order to format any text, you should select it first just as you would in Microsoft Word.

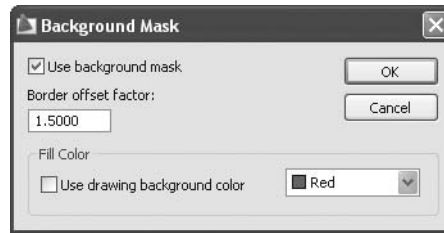
Formatting Panel

- The **Formatting** panel is as follows:



- Use the **Formatting** panel to change any or all of the following:
 - Change the text to be **Bold**.
 - Change the text to be **Italic**.
 - Change the text to be **Underlined**.
 - Change the text to be **Overlined**.

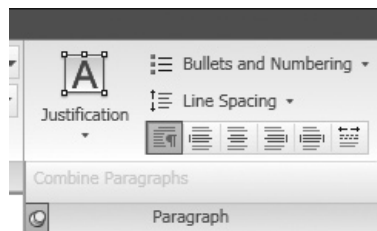
- Change the font (it is recommended to stick with the font specified by the current text style).
- Change the color of the text (it is recommended to stick with the color of the current layer).
- Convert an uppercase letter to a lowercase letter, and vice versa.
- Specify the **Background Mask** (the background color). You will see the following dialog box:



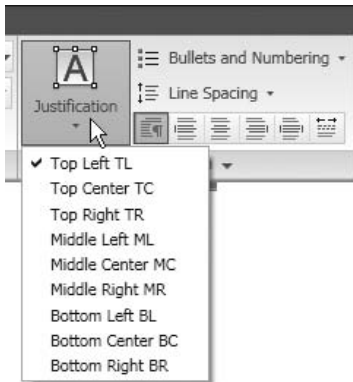
- Specify the **Oblique Angle**.
- Specify the **Tracking** to increase or decrease the spaces between letters. Values greater than 1 mean more space between letters, and vice versa.
- Specify the **Width Factor**.

Paragraph Panel

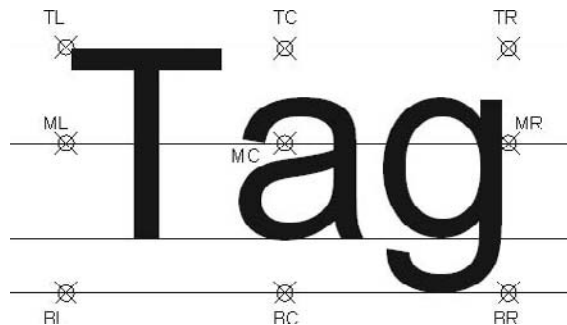
- The **Paragraph** panel is as follows:



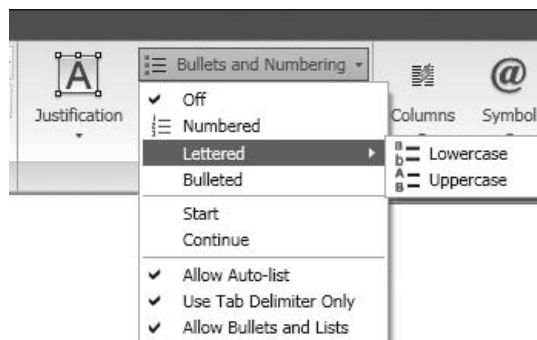
- Use the **Paragraph** panel to change any or all of the following: **Justification**, **Bullets and Numbering**, and **Line Spacing**.
 - Change the **Justification** to one of the following nine options:



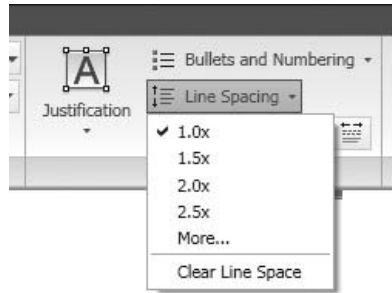
- To understand **Justification**, look at the following illustration:



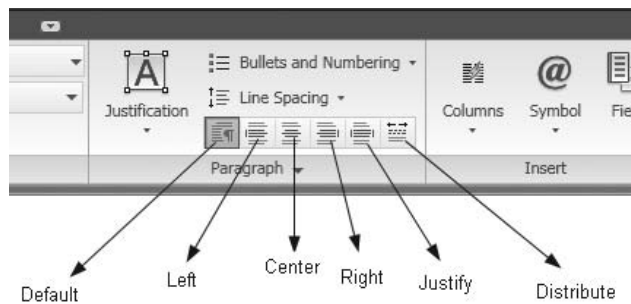
- You can use **Bullets and Numbering**, where you can choose between using letters (lowercase or uppercase), numbers, or bullets.



- You can set the **Line Spacing** of the paragraph. You have the choice of 1.0x, 1.5x, 2.0x, 2.5x, or you can set your own.

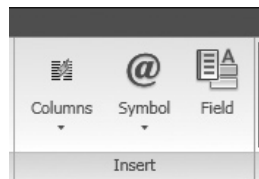


- You can change the justification of the paragraph by using the six justification buttons as illustrated:

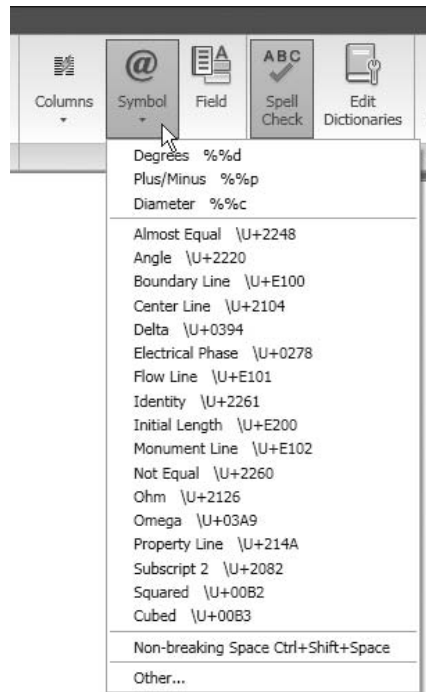


Insert Panel

- The **Insert** panel is as follows:

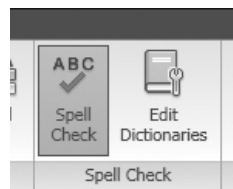


- You can add **Symbols** to your text. If you click the **Symbol** button, the following menu will appear. You can add one of 20 available symbols:



Spell Check Panel

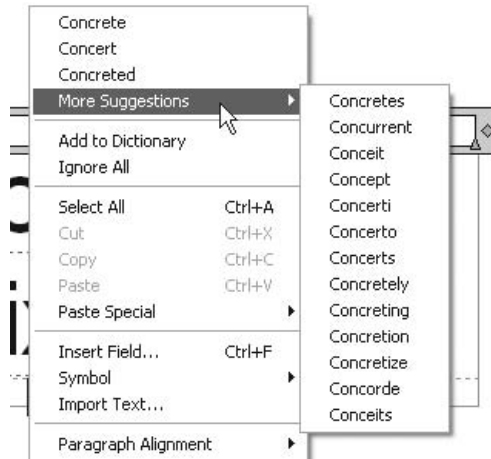
- The **Spell Check** panel is as follows:



- By default, the **Spell Check** button is on. While you are typing, a dotted red line will appear under the misspelled words, just like the following example:



- Go to the misspelled word and right-click. Suggestions for the correct spelling will appear at the top of the shortcut menu. If you need more suggestions, you can see them in a submenu, as shown in the following:



- To use other dictionaries, click the **Edit Dictionaries** button. You will see the following dialog box:

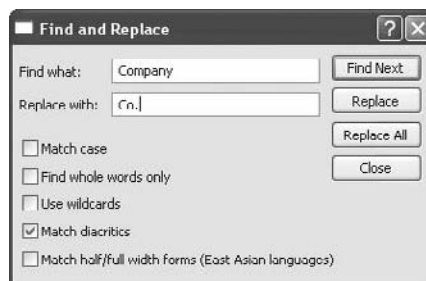


Tools Panel

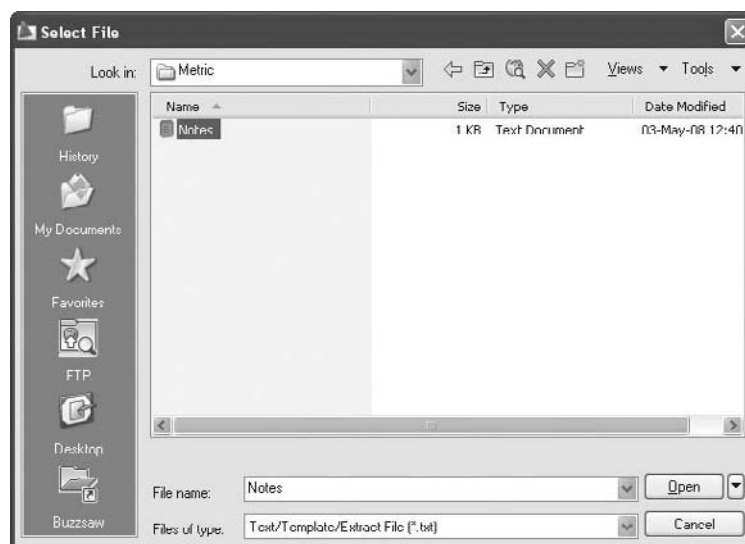
- The **Tools** panel is as follows:



- In the **Tools** panel, you can **Find and Replace** text, **Import Text**, and use the **AutoCAPS** button.
 - Click the **Find and Replace** button and the following dialog box will appear:



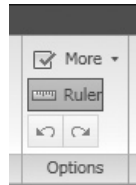
- You can replace one or all of the occurrences.
- You can use the **Import Text** button to place text already stored as a text file. The following dialog box will appear:



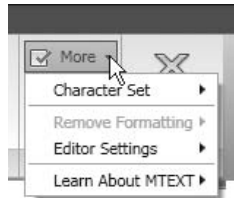
- Select the desired drive and folder, and then select the desired *.txt* file. Once you click **Open**, you will see the text in the editor available for formatting.
- Use the **AutoCAPS** button to write in uppercase letters (just like using **Caps Lock** on the keyboard).

Options Panel

- The **Options** panel is as follows:

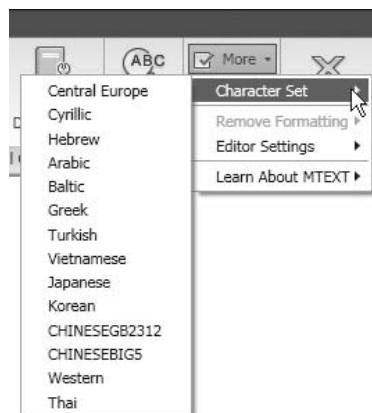


- Using the **Options** panel, you can do any or all of the following:
 - **Undo** any mistake you make while typing or **redo** it
 - **Show** or **Hide** the ruler
 - Show more options (see the following):



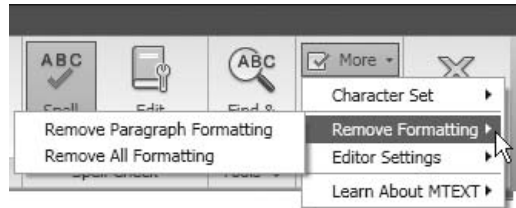
Character Set

- To choose a different character set, select from the submenu:



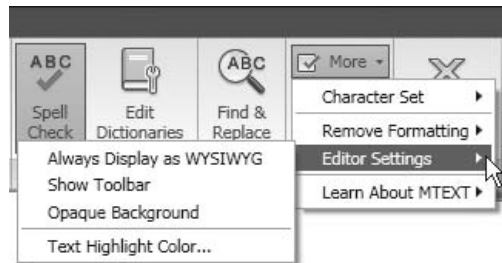
Remove Formatting

- In order for **Remove Formatting** to be active, you have to select the text first. AutoCAD will give you the ability to remove the formatting on the selected text:



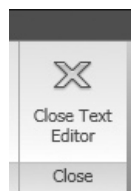
Editor Settings

- To control the **Editor Settings**, choose from the submenu:



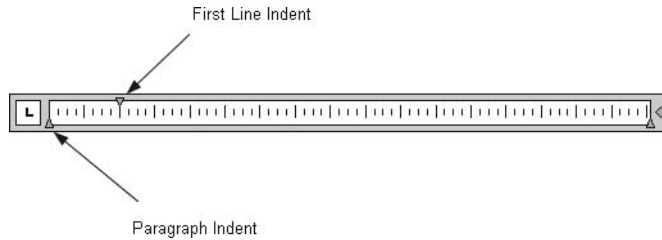
Close Panel

- The **Close** panel enables you to close the **Text Editor** tab:



Set Indents

- Setting indents in the **Text Editor** is identical to the process in Microsoft Word. See the following illustration:



- Move the **First Line Indent** to specify where the first line will start.
- Move the **Paragraph Indent** to specify where the next line will start.
- Also, you can set **Tabs** for your text by clicking anywhere in the ruler.

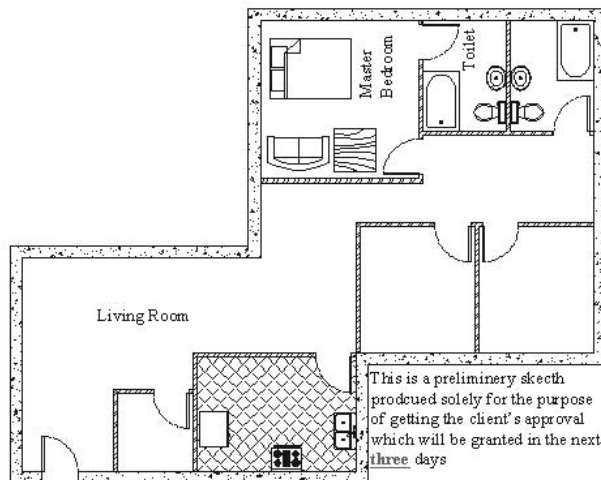
WRITING TEXT (METRIC AND IMPERIAL)



Workshops 13-A and 13-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_13.dwg*.
3. Review the following illustration:

Ground Floor Plan



4. Make the **Text** layer the current layer.
5. Using **Single Line Text** (**DTEXT** command) and the text style "Inside_Annot," type the following words: **Master Bedroom, Toilet, Living Room.**

6. Using **Single Line Text** (**DTEXT** command) and the text style “Title,” type the following words: **Ground Floor Plan**.
7. Switch off the **OSNAP** button.
8. Using the **Multiline Text** command, specify the area in the lower right-hand part of the plan as previously shown. Using the text style “Inside_Annot,” import the file named *Notes.txt*. Don’t close the text editor.
9. Select the word **three**, and make it red, bold, and underlined. Close the editor.
10. Save the file and close it.

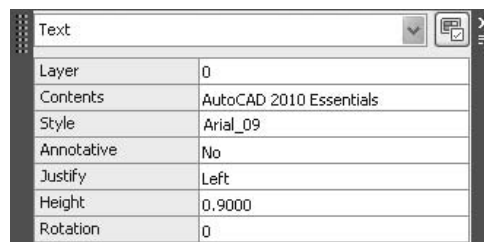
8.5 AN INTRODUCTION TO EDITING TEXT

- In order to edit the contents of the text, simply double-click the text.
- If you double-click multiline text, the editor will reappear with the **Multiline Text** tab. You can use this tab for further adding, deleting, or reformatting.
- If you double-click single line text, the text will be available for adding and deleting.
- Also, you can select multiline text, right-click, and select **Mtext Edit**.

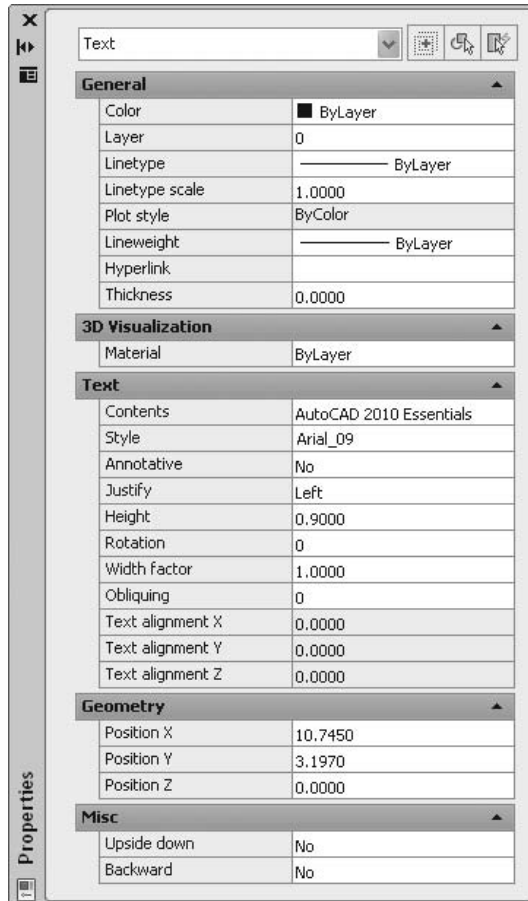
8.6 EDITING TEXT USING QUICK PROPERTIES AND PROPERTIES

Single Line Text

- In order to edit the properties of single line text, simply click it. The **Quick Properties** window will appear and you can make changes like the following:



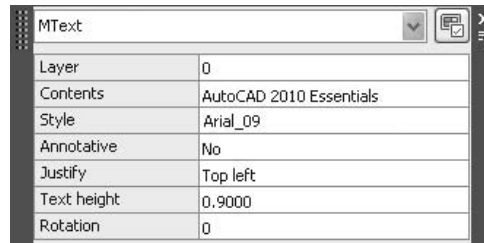
- You can change the **Layer**, **Contents**, **Style**, **Annotative** status, **Justification**, **Height**, and finally, the **Rotation** angle.
- If you want to do more editing, select **Properties**, and you will see the following:



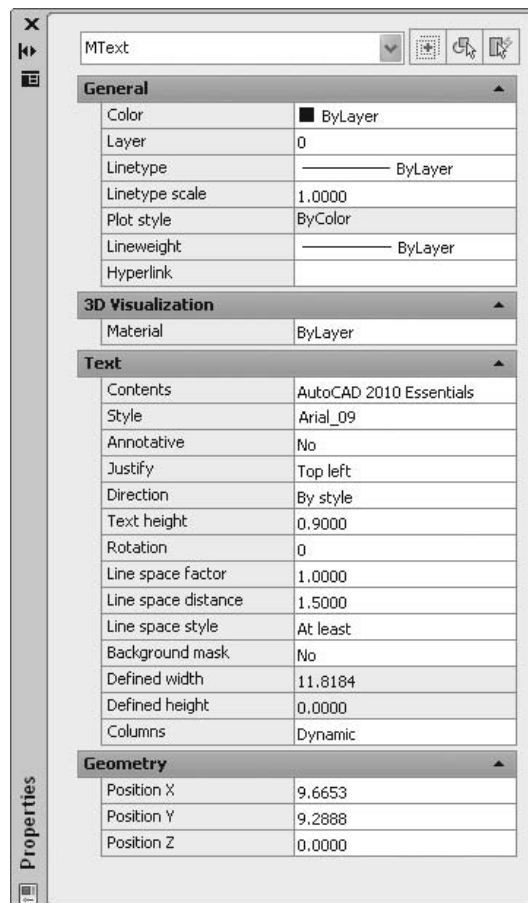
- You can change the **General** properties of the single line text (**Color**, **Layer**, **Linetype**, etc.).
- You can change the **Contents** of the text and other properties, such as **Style**, **Justification**, **Height**, **Rotation**, etc.).
- You can change the **Geometry** of the text (position of **X**, **Y**, and **Z**).
- Finally, you can change the **Miscellaneous** properties of single line text to **Upside down** and/or **Backward**.

Multiline Text

- In order to edit the properties of multiline text, simply click it. The **Quick Properties** window will appear and you can make changes as shown in the following:



- You can edit the same things you can change in single line text.
- In order to have full editing power, you need to start the **Properties** command. You will see the following:



- You can change the **General** properties of the multiline text (**Color**, **Layer**, **Linetype**, etc.)

- You can change the **Contents** of the multiline text and other properties like **Style**, **Justification**, **Direction**, **Height**, **Rotation**, etc.
- You can change the specific features of multiline text like the **Line space factor**, **Line space distance**, **Line space style**, **Background mask**, **Defined width**, and **Defined height**.
- If you select both single line text and multiline text, you can only change the **General** properties.
- You can select either multiple single line text or multiple multiline text and change their properties in one step.



8.7 TEXT AND GRIPS

- If you click on single line text, you will see the following:

Elevation

- The grip appears at the start point of the baseline.
- On the other hand, if you click on multiline text, you will see the following:

Elevations and Cross
Sections will be
modified together

- You will see a grip at the top left, which will allow you to move the multiline text.
- The arrow at the top right allows you to extend the horizontal dimension of the multiline text. If you stretch it to the right it will increase and if you stretch it to left it will decrease.

Elevations
and Cross
Sections
will be
modified
together

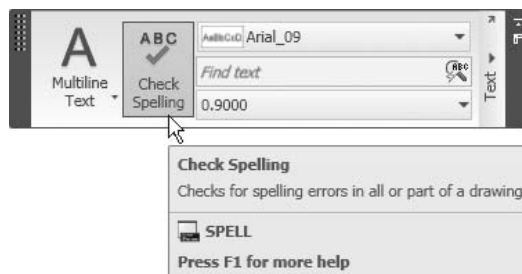
- The arrow at the bottom will place the text in multiple columns:

Elevations and Cross Sections will be modified together

8.8 CHECK SPELLING AND FIND AND REPLACE

Check Spelling

- We have already learned how to **Spell Check** and **Find and Replace** inside the multiline text editor. But what if we want to check an existing single line or multiline text? We can use two tools.
- AutoCAD will spell check the entire drawing, the current space/layout, or selected text.
- To start the **Check Spelling** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Text** panel, select the **Check Spelling** button.



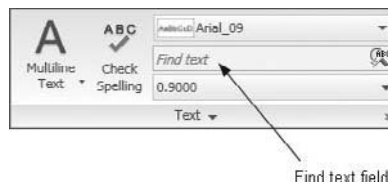
- The following dialog box will appear:



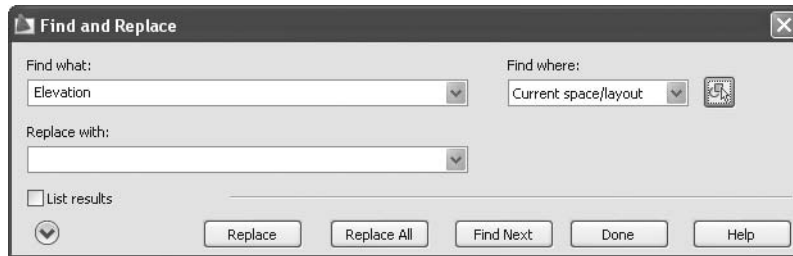
- This is identical to the spell checker in Microsoft Word. If AutoCAD finds a misspelled word, it will give you suggestions and you can choose one of them. You can also change or ignore the suggestion.

Find and Replace

- AutoCAD can find any word or part of a word in the entire drawing file and replace it.
- To start the **Find and Replace** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Text** panel, type the desired word in the **Find text** field:



- Type the word or phrase you are looking for in the edit box and click the key at the right. AutoCAD will locate the word and show the following dialog box:



- Under **Replace with**, type the new word(s) you want to appear in place of the word(s) given in **Find what**.
- You can search the entire drawing or search in a selection of text.
- You have three choices: **Find**, **Replace**, and **Replace All**.
- When you are finished, click **Done**.

EDITING TEXT (METRIC AND IMPERIAL)



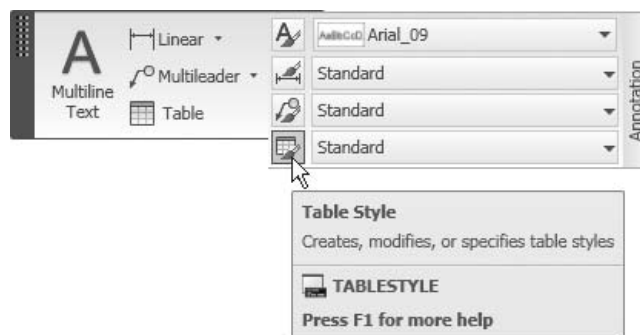
Workshops 14-A and 14-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_14.dwg*.

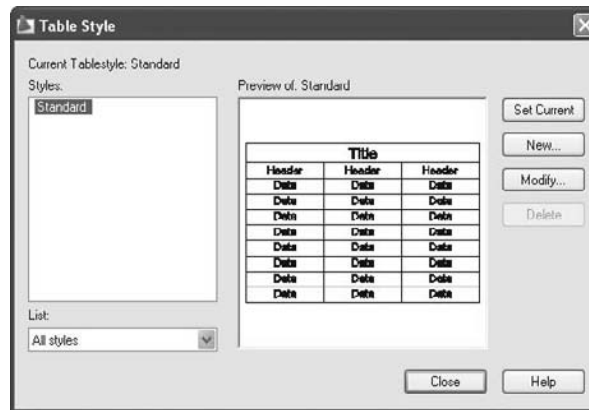
3. Select the multiline text and the four grips will appear. Select one of the right grips to make it hot and stretch it to the right to make the text one line less.
4. Double-click the multiline text and make the following changes:
 - a. Select the word “solely” and make it italic.
 - b. Add a comma before the word “which.”
 - c. Press [Enter] after the last word to add a new line and type your initials.
5. While you are in the text editor, you can see three words with dashed red lines beneath them. Use the **Check Spelling** command for these three words and select the correct spelling.
6. Save the file and close it.

8.9 TABLE STYLE

- To create a table in AutoCAD, perform the following two steps:
 - Create a table style.
 - **Insert** and **Fill** the table.
- In **Table Style** you will define the main features of your table.
- To start the **Table Style** command, make sure you are in the **Home** tab on the **Ribbon**, and, expanding the **Annotation** panel, click the **Table Style** button:



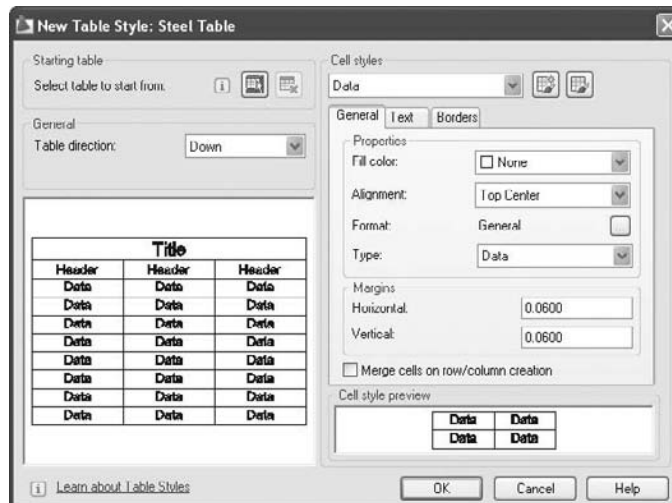
- You will see the following dialog box:



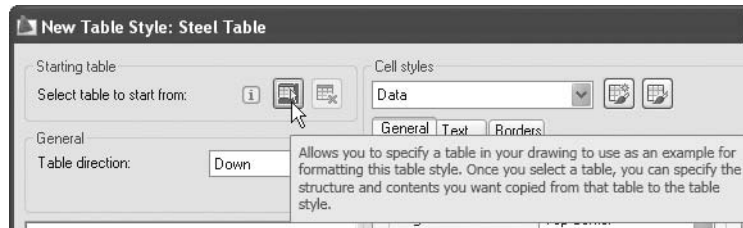
- As you can see, there is a predefined style called **Standard**.
- There will always be a preview that will show you the changes you are making. Therefore, it will be easy for you to decide if you have made the right choices.
- To create a new table style, click the **New** button. You will see the following dialog box:



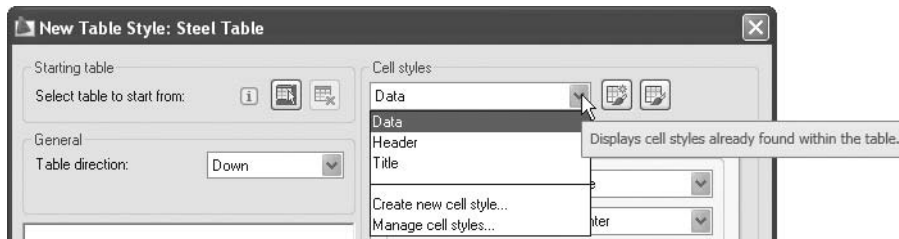
- Type in the name of your new style.
- Select the **Start With** style (you will start with a copy from this style).
- Click the **Continue** button. The following dialog box will appear:



- Under **Starting table**, you can select an existing table and copy its style instead of starting from scratch.



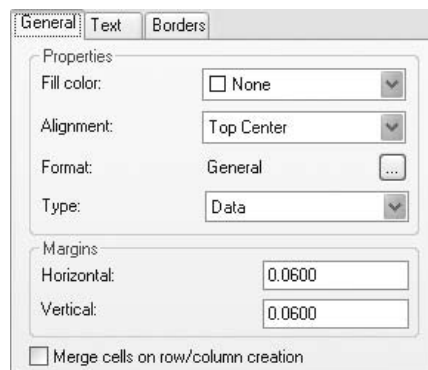
- Under **General**, specify the **Table direction**:
 - Down**: The title and column headers will be located at the top of the table and the cells will go below them.
 - Up**: The title and column headers will be located at the bottom of the table and the cells will go above them.
- Under **Cell styles**, you have three choices: **Data**, **Header**, and **Title**. This feature controls the table's three parts.



- You can control the **General** properties, **Text** properties, and **Border** properties of these three parts.

General Tab

- The **General** properties tab looks like the following:



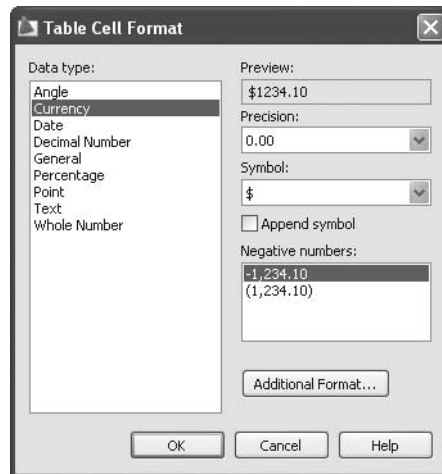
- Set the **Fill Color**, **Alignment**, **Format**, **Type**, and **Margins**.
 - **Fill Color**: Determine whether the cells will have a colored background.
 - **Alignment**: Determine the justification of the text compared to the cell (you have nine to choose from). To illustrate the last two options, see the following example:

Steel Table	
Dia	Length
#3	112
#4	147
#5	144

Fill Color = 9
Alignment = Middle Center

Fill Color = None
Alignment = Top Left

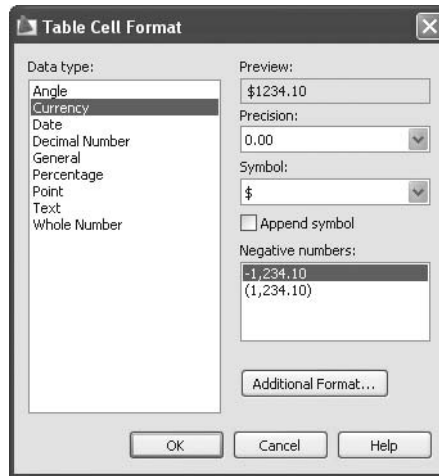
- **Format**: Select the format of the numbers. Click on the small button with the three dots and you will see the following dialog box:



- Set the **Type** to be **Data** or **Label**.
- **Margins**: Determine the **Horizontal** and **Vertical** distances to be taken around the **Data** relative to the borders.

Text Tab

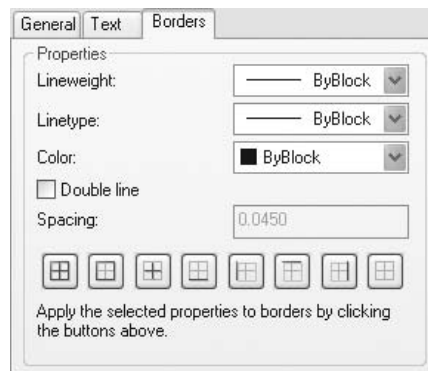
- The **Text** properties tab looks like the following:



- In the **Text** tab, you can control the following settings:
 - The text style you will use in the cells.
 - The text height to be used (this is only applicable if the selected text style has a Height = 0).
 - The text color to be used (you should most likely leave it **ByLayer** or **ByBlock**).
 - The text angle sets the oblique angle of the text.

Borders Tab

- The **Borders** properties tab looks like the following:



- In the **Borders** tab, you can control the following settings:
 - Specify the **Lineweight**, **Linetype**, and **Color** of the borders (either choose **ByBlock** or specify the desired value from the list).

- Specify whether you want the border to be single line (the default setting) or double line. If you specify double line, you must also specify the spacing.
- Set the type of border (inside, outside, etc.).
- Once you set all of the variables, click **OK** to go back to the first dialog box and do any or all of the following:
 - Select the current style from many available styles. The selected style becomes the default, which you will see the next time you use the **Table** command.
 - Select one of the existing styles and make any type of modification; you will see the same dialog box you see when you create a new style.

CREATING TABLE STYLE (METRIC)



Workshop 15-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_15.dwg*.
3. Create a new table style based on **Standard** and call it **Door Schedule**.
 - a. For title, **Text Style** = *Inside_Annot* and **Alignment** = Middle Center.
 - b. For data, **Text Style** = *Inside_Annot* and **Alignment** = Middle Center.
 - c. For header, **Text Style** = *Inside_Annot* and **Alignment** = Middle Center.
 - d. For cell margins, **Horizontal** = 100 and **Vertical** = 100.
4. Make it current.
5. Save the file and close it.

CREATING TABLE STYLE (IMPERIAL)

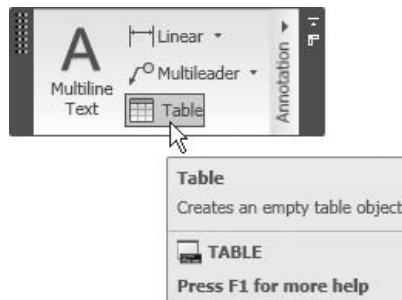


Workshop 15-B

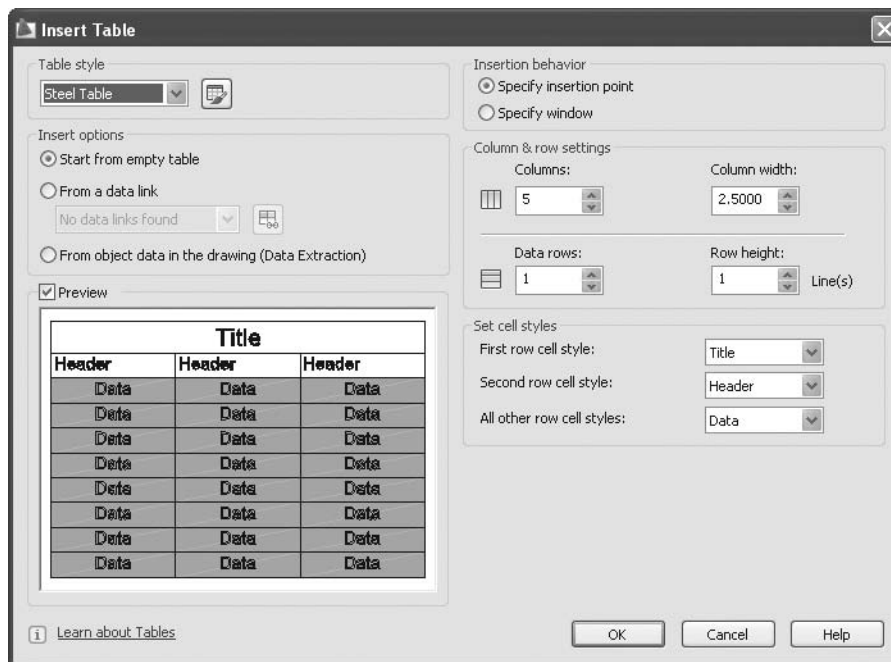
1. Start AutoCAD 2010.
2. Open the file *Workshop_15.dwg*.
3. Create a new table style based on **Standard** and call it **Door Schedule**.
 - a. For title, **Text Style** = *Inside_Annot* and **Alignment** = Middle Center.
 - b. For data, **Text Style** = *Inside_Annot* and **Alignment** = Middle Center.
 - c. For header, **Text Style** = *Inside_Annot* and **Alignment** = Middle Center.
 - d. For cell margins, **Horizontal** = 4" and **Vertical** = 4"
4. Make it current.
5. Save the file and close it.

8.10 THE TABLE COMMAND

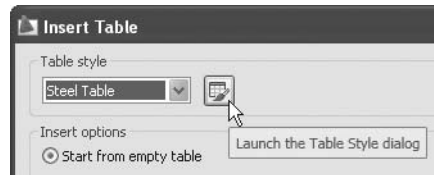
- Use the **Table** command to insert a table in an AutoCAD drawing using a predefined style.
- You will specify the number of columns and rows, and fill the cells with the desired data.
- To start the **Table** command, make sure you are in the **Home** tab on the **Ribbon**, and, using the **Annotation** panel, click the **Table** button.



- The following dialog box will appear:



- Select the predefined **Table style** name.
- If you did not create a table style before this step, simply click the small button next to the list, so you can start creating the desired table style.



- Specify the **Insert options**. You have three choices:
 - Start from an empty table—normally, this option should be used.
 - Import from a data link—to bring in data from spreadsheets created in an application like Microsoft® Excel.
 - Import from object data in the drawing (**Data Extraction**)—use only if you have block attributes.
- There are two insertion methods:
 - Specify insertion point
 - Specify window

Specify Insertion Point

- If you use this method, you will need to specify the upper left-hand corner of the table and set up the following data accordingly:
 - Number of columns
 - Column width
 - Number of rows (without title and column heads)
 - Row height (in lines)
- Click **OK** and AutoCAD will prompt:

Specify insertion point:

- Specify the upper left-hand corner of the table and the table will appear ready to fill the data in each row.
- First you will fill the title, then the column headers, and then the data. You can move between rows using the [Tab] key to go to the next cell and the [Shift]+[Tab] keys to go back to the previous cell.

Specify Window

- If you use this method, you will be asked later to specify a window. You must specify a total height and a total width.

- If you specify the number of columns, the column width will be calculated automatically (total width/number of columns). Or, if you specify the column width, the number of columns will be calculated automatically (total width/single column width).
- The same thing applies to rows. Either specify the number of rows, and the row height will be calculated automatically (total height/number of rows), or specify the row height, and the number of rows will be calculated automatically (total height/single row height).
- Click **OK** and AutoCAD will prompt:

Specify first corner:

Specify second corner:

- Specify two opposite corners and the table will be available for input just like the previous method.
- To edit cell content, simply double-click the cell and it will be available for editing.



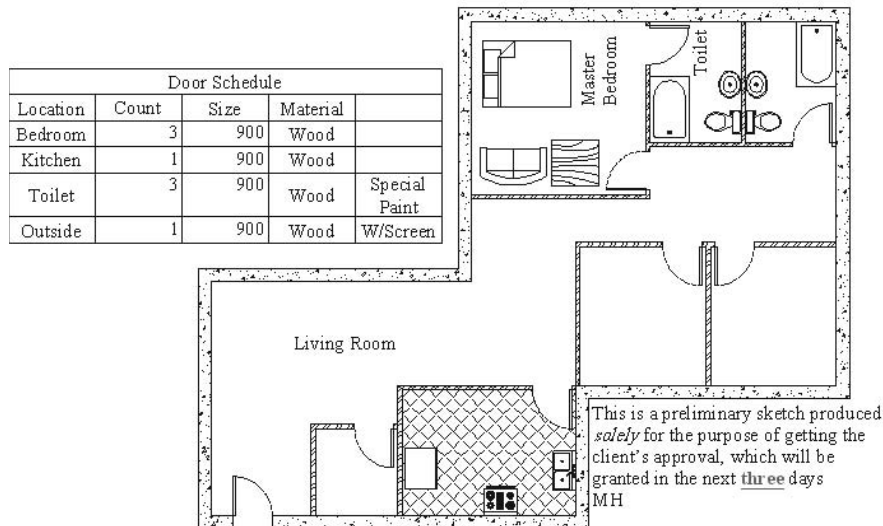
INSERTING TABLES (METRIC)



Workshop 16-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_16.dwg*.
3. Make the **Text** layer the current layer.
4. Look at the following illustration. Using the **Door Schedule** table style, add a table just like the one here, using the following:
 - a. Specify insertion point
 - b. **Columns** = 5
 - c. **Column Width** = 2000
 - d. **Data Rows** = 4
 - e. **Row Height** = 1 Line(s)

Ground Floor Plan



5. Save the file and close it.

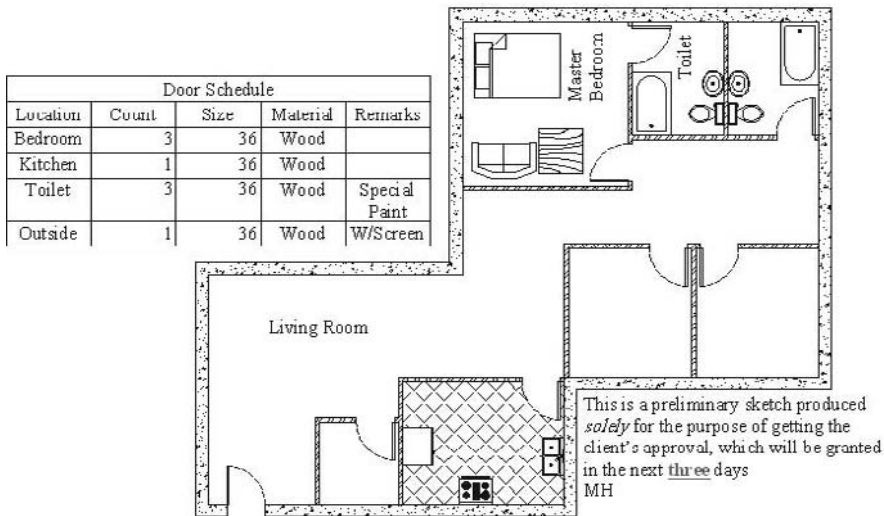
INSERTING TABLES (IMPERIAL)



Workshop 16-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_16.dwg*.
3. Make the **Text** layer the current layer.
4. Look at the following illustration. Using the **Door Schedule** table style, add a table just like the one here, using the following:
 - a. Specify insertion point.
 - b. **Columns** = 5
 - c. **Column Width** = 6'-8"
 - d. **Data Rows** = 4
 - e. **Row Height** = 1 Line(s)

Ground Floor Plan



5. Save the file and close it.

CHAPTER REVIEW

1. The height mentioned in the **Text Style** is for lowercase letters.
 - a. True
 - b. False
2. There are two types of fonts in AutoCAD: _____ and _____.
3. There is no relation between **Text style** and **Table style**.
 - a. True
 - b. False
4. While you are in the **multiline text editor** you can't:
 - a. Import any *.txt* files.
 - b. Format text.
 - c. Change the indents.
 - d. Bring in a Microsoft Word document as OLE.
5. Which one of the following statements is not true about tables?
 - a. There are two methods to insert a table.
 - b. You can control the cell style for title, header, and data.

- c. You can convert multiline text to a table.
- d. You can define the table direction whether it is down or top.
- 6. In **Table style**, Top, Left, and Middle are considered to be _____ options.

CHAPTER REVIEW ANSWERS

- 1. b
- 2. *.shx, .ttf*
- 3. b
- 4. d
- 5. c
- 6. **Justification**

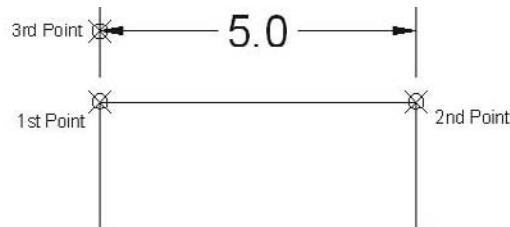
Chapter 9 DIMENSIONING YOUR DRAWING

In This Chapter

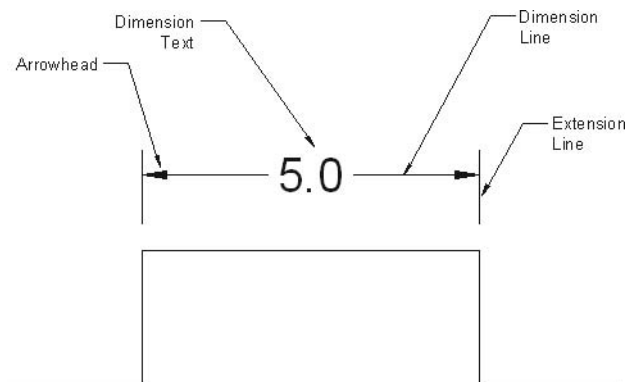
- ◇ Introduction
- ◇ Dimension Types
- ◇ **Dimension Style:** The First Step
- ◇ The **Lines** Tab
- ◇ The **Symbols and Arrows** Tab
- ◇ The **Text** Tab
- ◇ The **Fit** Tab
- ◇ The **Primary Units** Tab
- ◇ The **Alternate Units** Tab
- ◇ The **Tolerances** Tab
- ◇ Creating a Sub Style
- ◇ Controlling **Dimension Styles**
- ◇ An Introduction to Dimensioning Commands
- ◇ The **Linear** Command
- ◇ The **Aligned** Command
- ◇ The **Angular** Command
- ◇ The **Arc Length** Command
- ◇ The **Radius** Command
- ◇ The **Diameter** Command
- ◇ The **Jogged** Command
- ◇ The **Ordinate** Command
- ◇ The **Continue** Command
- ◇ The **Baseline** Command
- ◇ The **Quick Dimension** Command
- ◇ Dimension Blocks and Grips
- ◇ Dimension Block Properties
- ◇ An Introduction to the Multileader
- ◇ Multileader: Creating the Style
- ◇ **Multileader** Commands

9.1 INTRODUCTION

- Dimensioning in AutoCAD® is a semi-automatic process; users contribute part of the job and AutoCAD does the rest.
- Users in linear dimensioning, for example, specify three points. The first and the second is the length to be dimensioned, and the third is the position of the dimension line.
- Accordingly, AutoCAD will generate the **Dimension block** automatically, as shown in the following illustration:



- A **Dimension block** consists of four parts. They are:
 - Dimension line
 - Extension lines
 - Arrowheads
 - Dimension text
- See the following illustration:



- The Dimensioning process has two phases:
 - Creating dimension style(s)
 - Dimensioning your drawing
- The dimension style will control the appearance of the **Dimension block**. Each user will set up the style according to their needs.

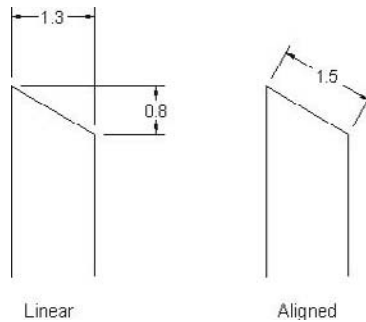
- Creating a dimension style is a lengthy and tedious job, but it will be done only once, thereby allowing users to focus on the other job of putting dimensions over the drawing.

9.2 DIMENSION TYPES

- AutoCAD can support the following dimension types:
 - Linear and Aligned
 - Arc Length, Radius, and Diameter
 - Angular
 - Continuous
 - Baseline
 - Ordinate

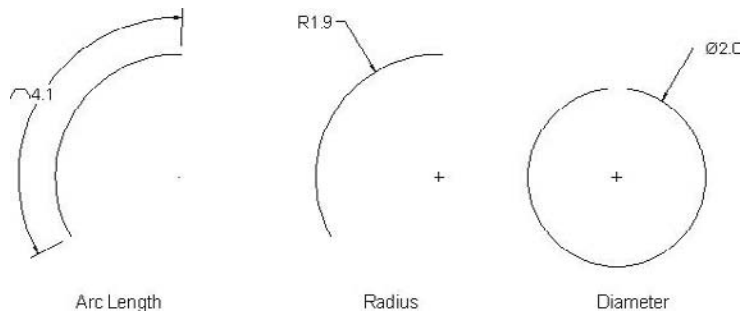
Linear and Aligned

- See the following illustration:



Arc Length, Radius, and Diameter

- See the following illustration:



Angular

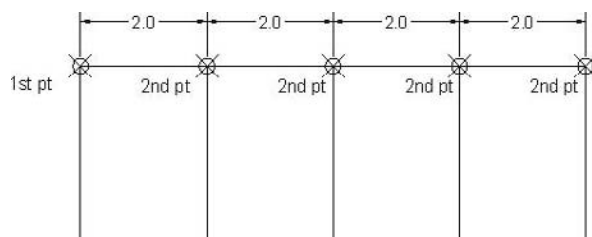
- See the following illustration:



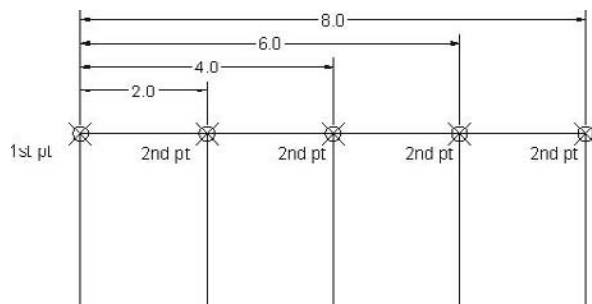
Angular

Continuous

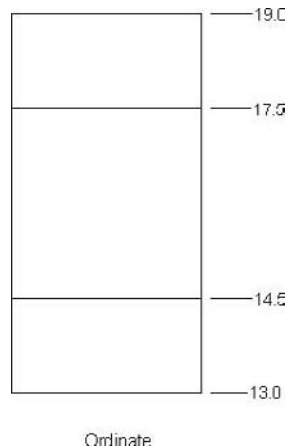
- See the following illustration:

**Baseline**

- See the following illustration:

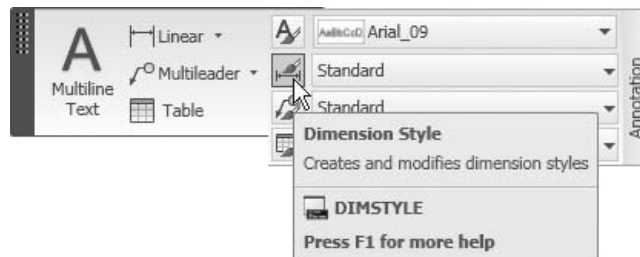
**Ordinate**

- See the following illustration:

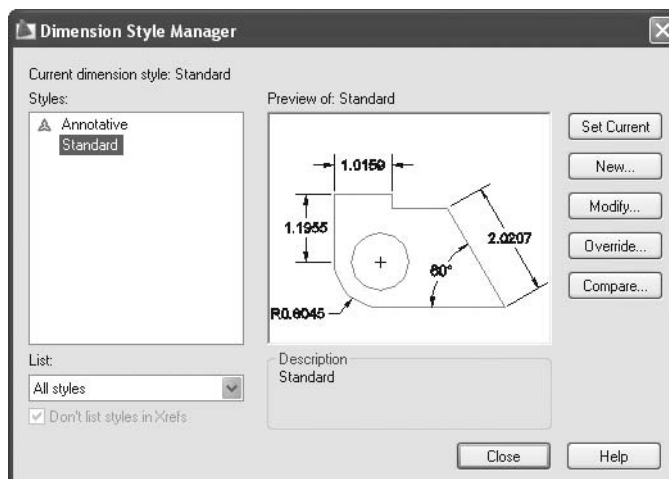


9.3 DIMENSION STYLE: THE FIRST STEP

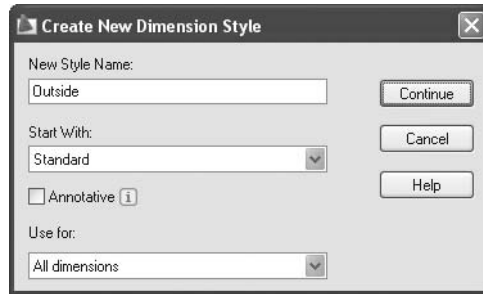
- To start the **Dimension Style** command, make sure you are in the **Home** tab on the **Ribbon**, and, expanding the **Annotation** panel, click the **Dimension Style** button.



- The following dialog box will appear:



- By default, there is a template for the dimension style called **Standard**.
- You can modify this style or create your own (which is the more preferable choice).
- To create a new style, click the **New** button. The following dialog box will appear:

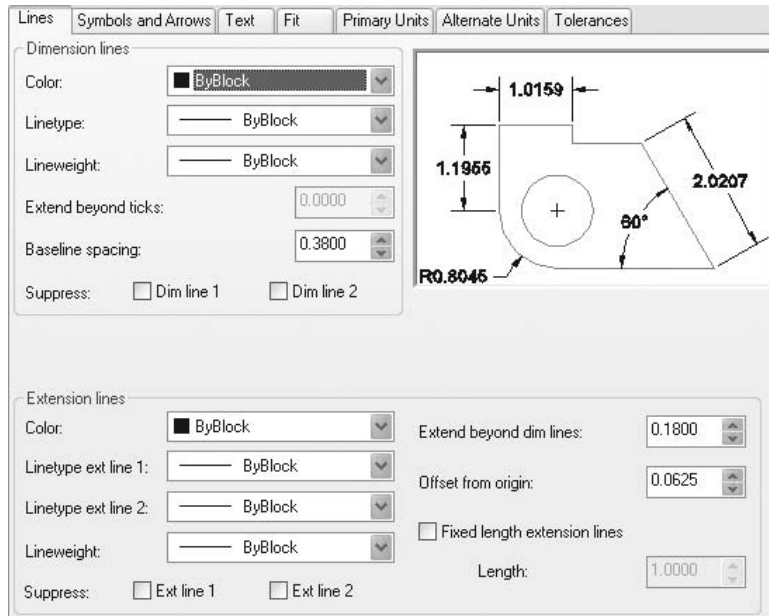


- Type in a new style name using the same naming convention you used when creating layers.
- Select the **Start With** style (you will start with a copy of this style).
- Keep **Annotative** off for now.
- By default, the changes you will make will be implemented for all types of dimensions. However, you can also create a new dimension style that will affect a certain type of dimension.
- Click the **Continue** button and begin modifying the settings. We will cover each tab of the **Dimension Style** dialog box in the coming pages.
- Whenever you find a **Color** setting, leave it as is. It is preferable to control colors through layers and not through individual objects. This also applies for **Linetype** and **Lineweight**.

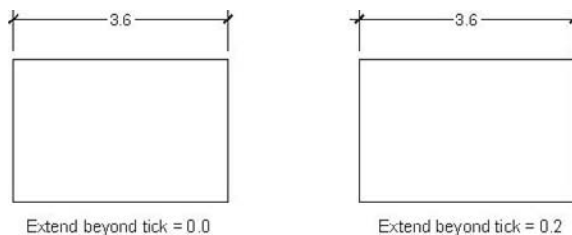


9.4 THE LINES TAB

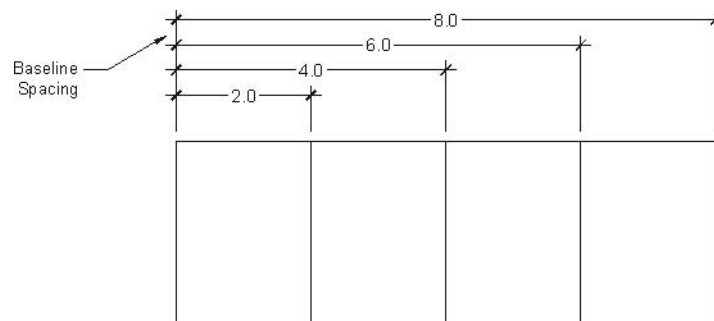
- The first tab in the **Dimension Style** dialog box is **Lines**. This is where we will control **Dimension lines** and **Extension lines**. It looks like the following:



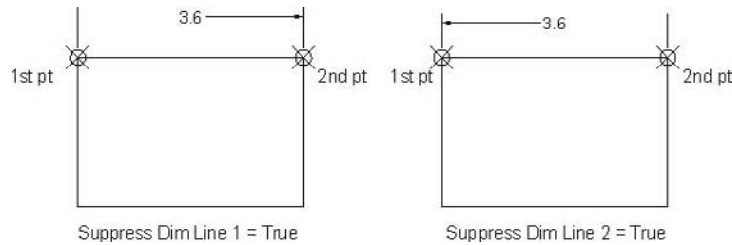
- You can control the following settings under **Dimension lines**:
 - The **Color**, **Linetype**, and **Lineweight** of the dimension line.
 - **Extend beyond ticks** (to edit this value, go to the **Symbols and Arrows** tab and set the **Arrowhead** to be either **Architectural tick** or **Oblique**). See the following illustration:



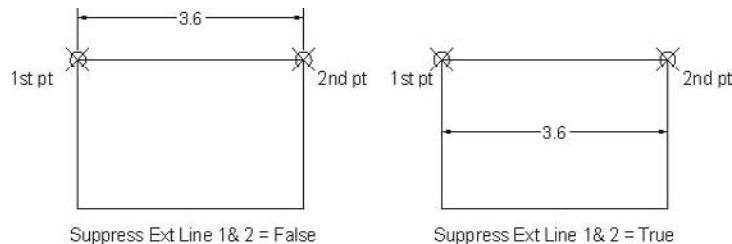
- See the following illustration for **Baseline spacing**:



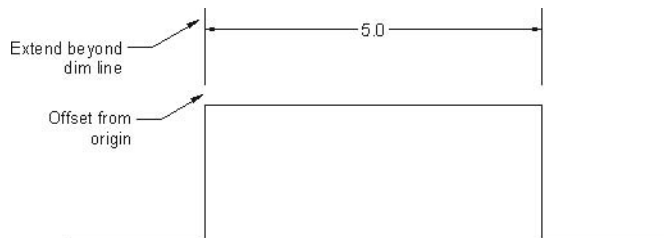
- Select to **Suppress Dim line 1** for one of them or **Dim line 2** for both:



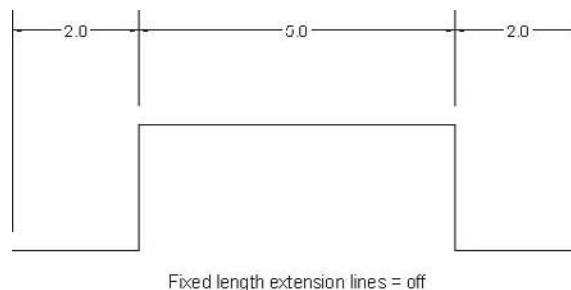
- Under **Extension lines**, you can control the following settings:
 - The **Color**, **Linetype**, and **Lineweight** of the extension lines.
 - Select to **Suppress Ext line 1** for one of them or **Ext line 2** for both:

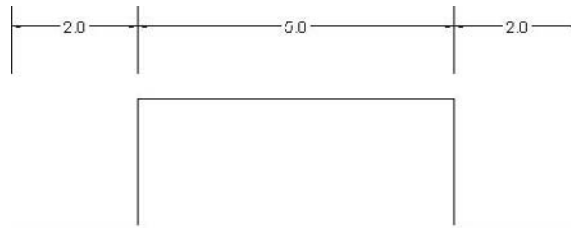


- Specify **Extend beyond dim lines** and **Offset from origin**:



- In order to set **Fixed length extension lines**, you have to specify the **Length**. See the following example for a better understanding:





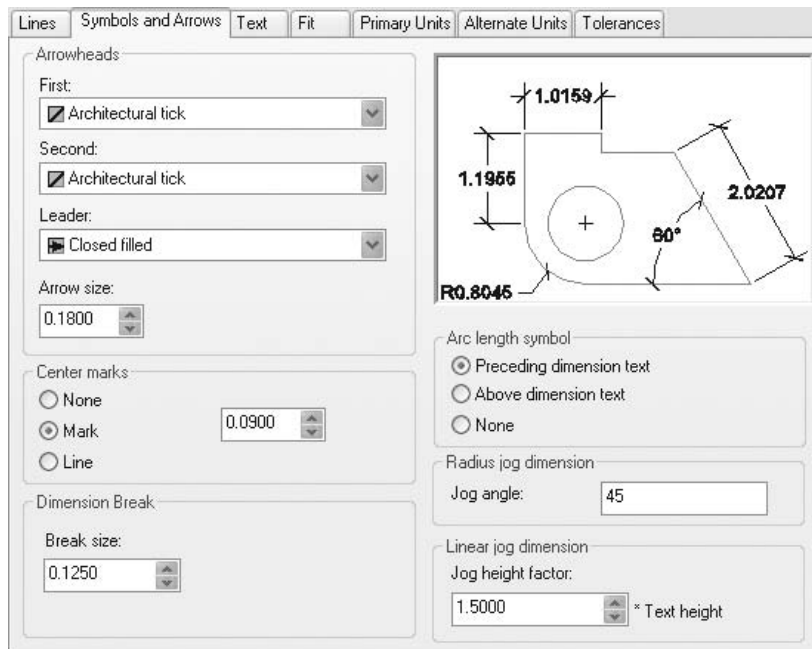
Fixed length extension lines = on & Length = 1.00



- The **Length** you specify will be calculated from the dimension origin up to the dimension line.

9.5 THE SYMBOLS AND ARROWS TAB

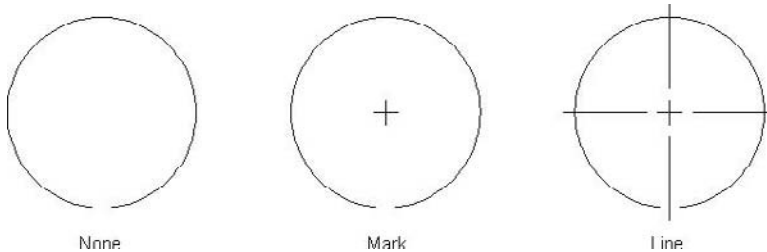
- When you click on the **Symbols and Arrows** tab, you will see the following:



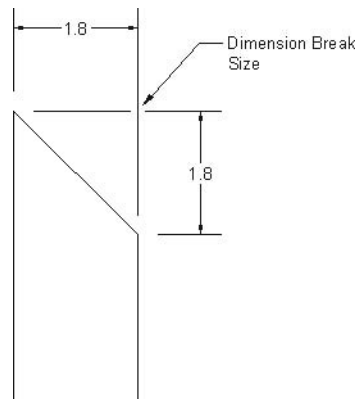
- Under **Arrowheads**, you can control the following settings:
 - The shape of the **First** arrowhead.
 - The shape of the **Second** arrowhead.
 - The shape of the arrowhead to be used in the **Leader**.
 - The **Size** of the arrowhead.



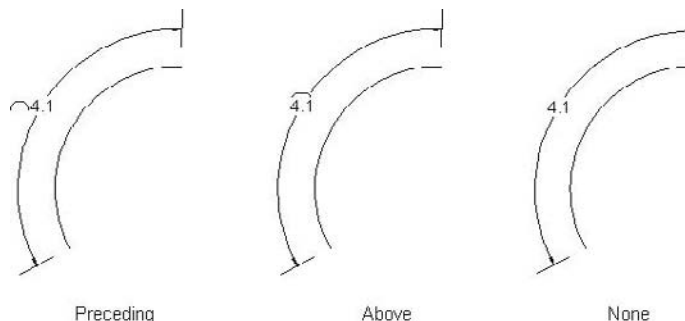
- If you change the first, the second will change automatically, but if you change the second, the first will not change.
- Under **Center marks**, you can:
 - Select whether or not to show the center mark or the centerlines.
 - Set the **Size** of the center mark.



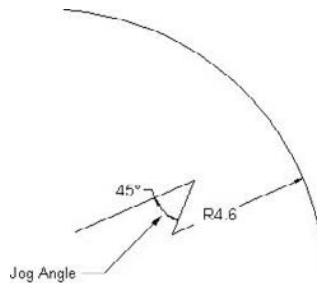
- Under **Dimension Break**, set the **Break size**. This is the width of the break of the dimension lines in a dimension break.



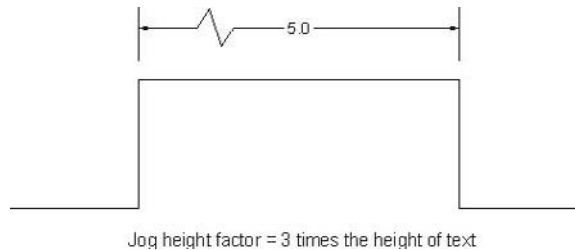
- Under **Arc length symbol**, you can decide whether to show the symbol **Preceding** the dimension text, **Above** the dimension text, or not at all (**None**). See the following illustration:



- Under **Radius jog dimension**, set the value of the **Jog angle**:

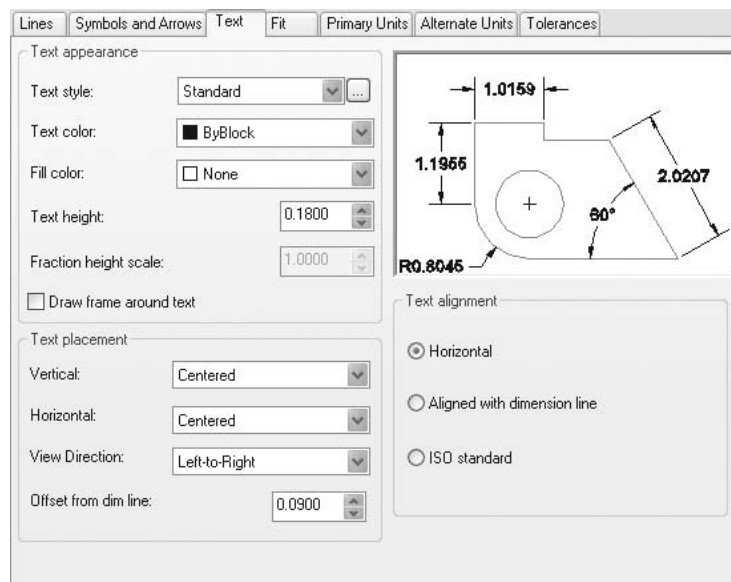


- Under **Linear jog dimension**, set the **Jog height factor** as a percentage of the text height. See the following illustration:

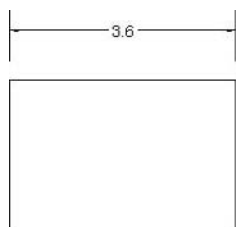


9.6 THE TEXT TAB

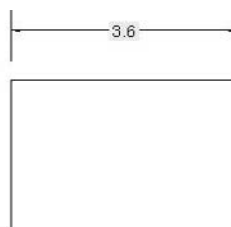
- When you click on the **Text** tab, you will see the following:



- Under **Text appearance**, you can control the following settings by:
 - Selecting the desired premade **Text style** to be used to input the dimension text. If you did not create a text style prior to this step, you can click the button with the three dots and create it now.
 - Specifying the **Text color**.
 - Specifying the **Fill color** (the background color).

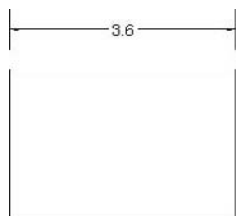


Fill color = None

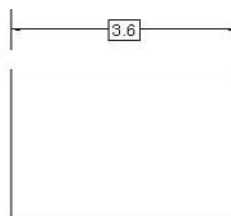


Fill color = Yellow

- Inputting the **Text height** (only applicable if the assigned text style has a text height = 0.0).
- Going to the **Primary Units** tab and assigning the **Unit format** to be **Architectural** or **Fractional**. Then, the dimension text will appear close to 1 1/4. You can set the **Fraction height scale** if you would like the text set at a different height.
- Deciding whether or not to **Draw frame around** the dimension text.



Draw frame around text = off



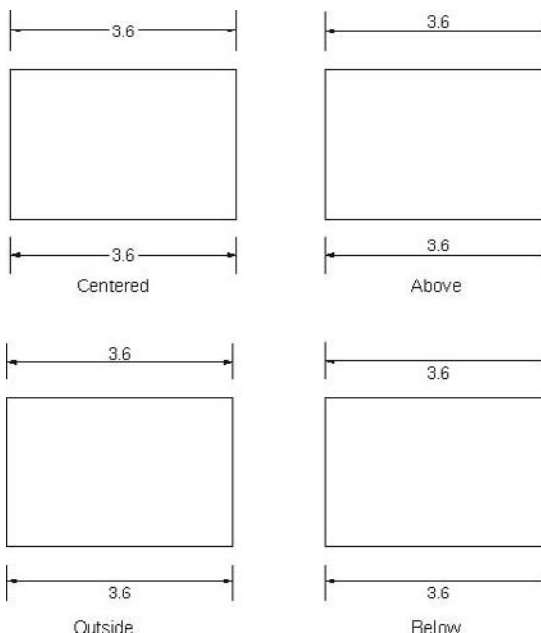
Draw frame around text = on

- As an introduction to **Text placement**, see the following illustration:

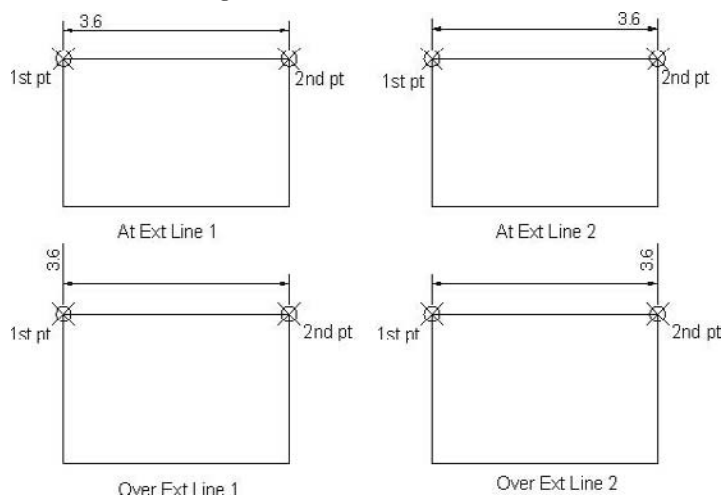


Vertical placement = Centered
Horizontal placement = Centered

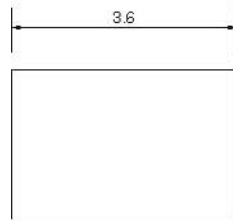
- Under **Text placement**, you can:
 - Select the **Vertical** placement. You have five choices: **Centered**, **Above**, **Outside**, **JIS** (Japan Industrial Standard), and **Below**. See the following illustration:



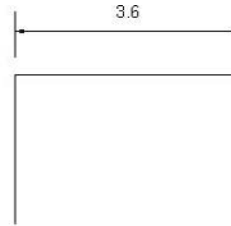
- Select the **Horizontal** placement. You have five choices: **Centered**, **At Ext Line 1**, **At Ext Line 2**, **Over Ext Line 1**, and **Over Ext Line 2**. See the following illustration:



- Set the **View Direction** of the dimension text to be either **Left-to-Right** or **Right-to-Left**.
- Set the **Offset from dim line**, which is the distance between the dimension line and the baseline of the dimension text. See the following illustration:

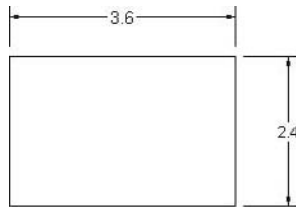


Offset from dim line = 0.00

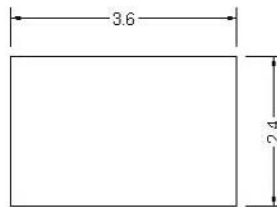


Offset from dim line = 0.30

- Under **Text alignment**, control whether the text will always be **Horizontal**, **Aligned with dimension line**, or according to **ISO standard**. See the following illustration:



Text alignment = Horizontal



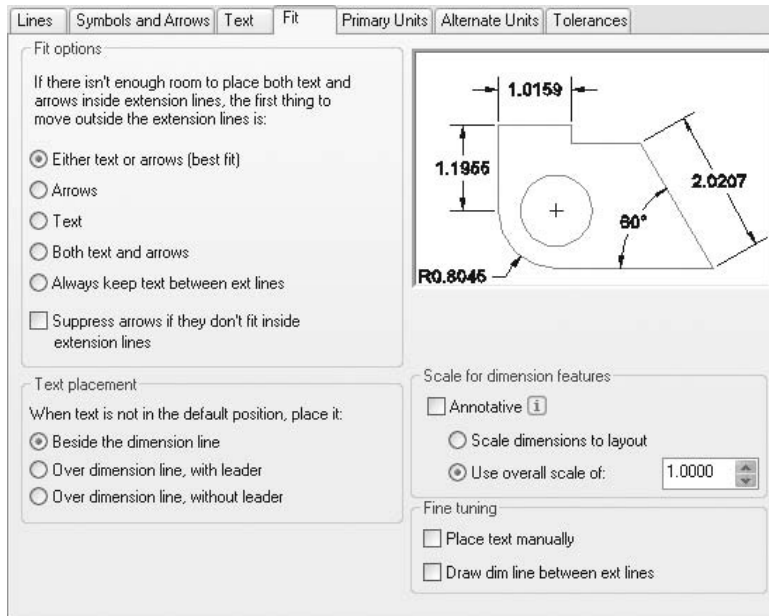
Text alignment = Aligned with dimension line



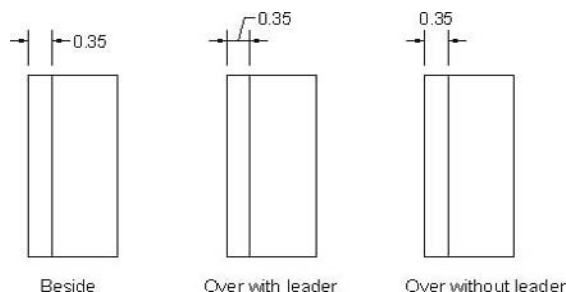
- The only difference between **Aligned with dimension line** and **ISO standard** is the **Radius** and **Diameter** types. The first is considered, like the other types, to be aligned with the dimension line, and **ISO standard** is considered to be horizontal.

9.7 THE FIT TAB

- When you click on the **Fit** tab, you will see the following:



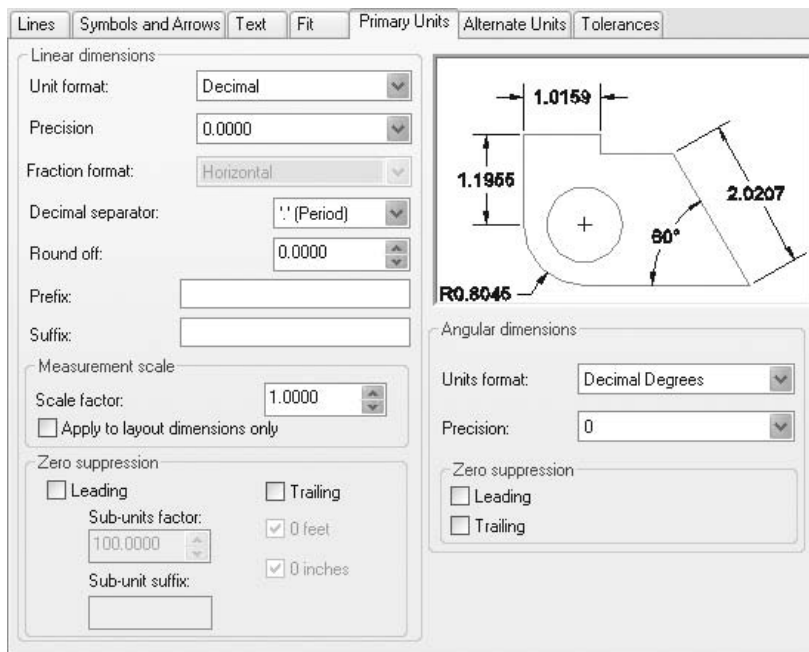
- As an introduction to the **Fit** tab, notice that there are three things that will be within the two extension lines: the **Dimension line**, **Arrowheads**, and the **Dimension text**. AutoCAD will put them all inside the two extension lines when the distance can accommodate all three. What if there is not enough room? We will discuss that option here.
- Under **Fit options**, adjust the following settings:
 - Select one of the five options to decide how AutoCAD will treat **Arrowheads** and **Dimension text**.
 - Select whether to **Suppress arrows if they don't fit inside extension lines** or not.
- Under **Text placement**, control the placement of the text. If it does not fit inside the extension lines, you have three options: **Beside the dimension line**, **Over dimension line with leader**, and **Over dimension line without leader**. See the following illustration:



- Under **Scale for dimension features**, adjust the following:
 - Keep **Annotative** off for now.
 - Set the **Scale dimension to layout** (we will discuss layouts in the next chapter).
 - For any distance, length, or size, you will input a value. **Use overall scale of** is a setting that will magnify or shrink whole values in one step. This will not affect the distance measured.
- Under **Fine tuning**:
 - If you do not trust AutoCAD to place your text in the right place, you can choose to **Place text manually**.
 - Also, you can choose to force AutoCAD to draw the dimension line between the extension lines by choosing **Draw dim line between ext lines**, whether the distance is appropriate or not.

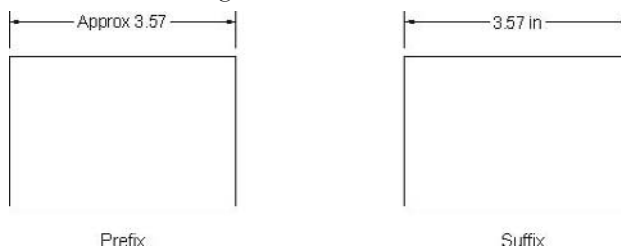
9.8 THE PRIMARY UNITS TAB

- When you click on the **Primary Units** tab, you will see the following:

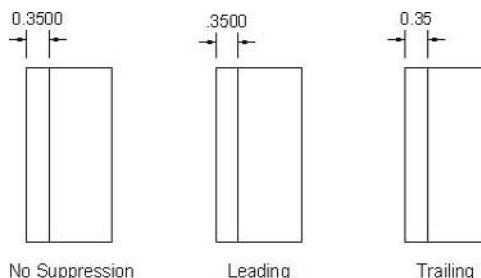


- As an introduction to the **Primary Units**, let us assume that your client wants the dimensions in Decimal format and a subcontractor wants it in Architectural format. The solution will be to show two numbers for each dimension, where the first will be the **Primary Units** and the second will be the **Alternate Units**. In this section we will cover the **Primary Units**.

- Under **Linear dimensions**:
 - Choose the **Unit format** and select one of the six available formats.
 - Select the **Precision** of the unit format selected.
 - If you select the Architectural or Fractional format, specify the **Fraction format**, which includes **Horizontal**, **Diagonal**, and **Not Stacked**.
 - If you select **Decimal**, specify the **Decimal Separator**—**Period**, **Comma**, or **Space**.
 - Specify the **Round off** number. If you select 0.5, for example, then AutoCAD will round off any dimension to the nearest 0.5.
 - Input the **Prefix** and/or the **Suffix**. See the following illustration for a better understanding.



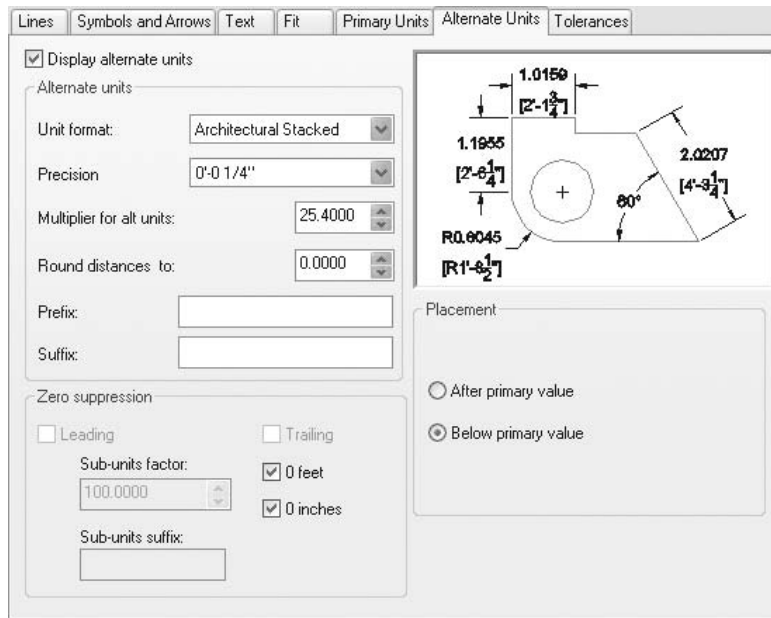
- Under **Measurement scale**:
 - Input the **Scale factor**. To understand the importance of this setting, let's assume we have a drawing that uses the millimeter as the unit—hence a length of 10 m will be 10,000—but we want the value 10 to appear and not 10,000. To do so, we set the **Scale factor** to be 0.001.
 - Select **Apply to layout dimensions only** (we will discuss the layouts in the next chapter).
- Under **Zero suppression**, select to suppress the **Leading**, and/or the **Trailing** zeros. See the following illustration:



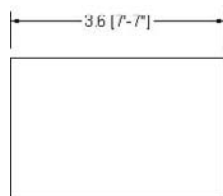
- If you have meters as your unit, and the measured value is less than one centimeter, we call this a sub unit. Set up the **Sub-units factor** (in our example 1 m = 100 cm, so the factor = 100) and the suffix for it.
- Under **Angular dimensions**, select the **Unit format** and the **Precision**.
- Under **Zero suppression**, select to suppress the **Leading** and/or the **Trailing** zeros for the angular measurements.

9.9 THE ALTERNATE UNITS TAB

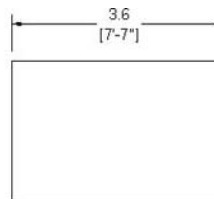
- When you click on the **Alternate Units** tab, you will see the following:



- If you want two numbers to appear for each dimension, click on **Display alternate units**.
- Specify the alternate **Unit format**, its **Precision**, the **Multiplier for all units** value, the **Round distances to**, the **Prefix**, the **Suffix**, and the **Zero suppression** criteria.
- Specify whether to show the alternate units **After primary value** or **Below primary value**. See the following illustration:



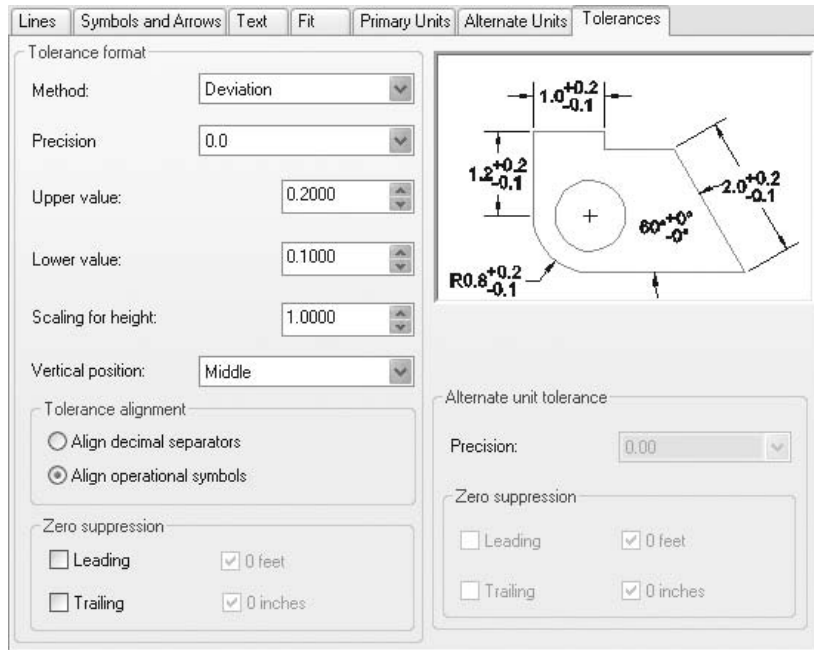
After primary value



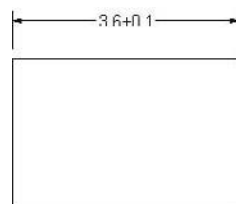
Below primary value

9.10 THE TOLERANCES TAB

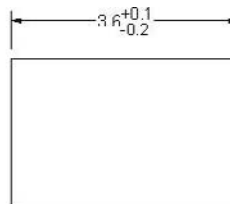
- When you click on the **Tolerances** tab, you will see the following:



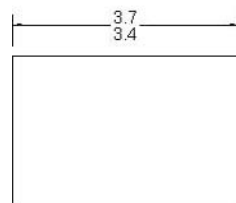
- There are several ways to show the tolerances, they are:
 - **Symmetrical**
 - **Deviation**
 - **Limits**
 - **Basic**
- See the following illustration:



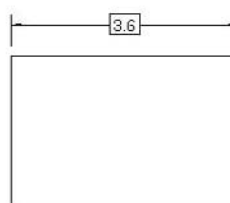
Symmetrical



Deviation

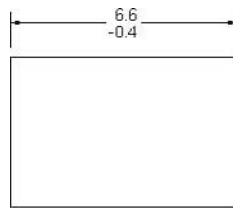


Limits

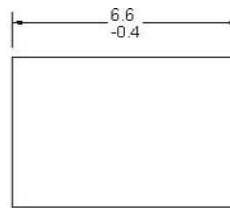


Basic

- Under the **Tolerance format**:
 - Specify the desired **Method** from the preceding list.
 - Specify the **Precision** of the numbers to be shown.
 - If you select **Symmetrical**, specify the **Upper value**.
 - For **Deviation** and **Limits**, specify the **Upper value** and **Lower value**.
 - If you want the tolerance values to appear smaller than the dimension text, specify **Scaling for height**.
 - Specify the **Vertical position** of the dimension text with regard to the tolerance values (**Bottom**, **Middle**, or **Top**).
- If **Deviation** or **Limits** is selected, choose whether to **Align decimal separators** or **Align operational symbols**. See the following illustration:



Align decimal separators

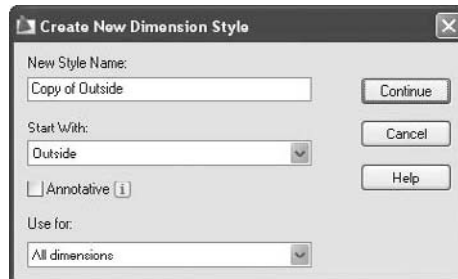


Align operational symbols

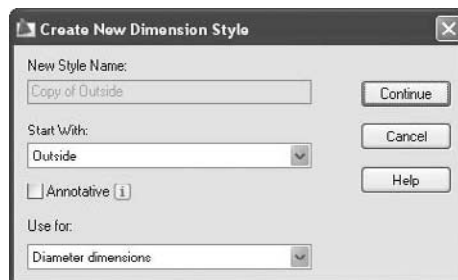
- If you are showing **Alternate units**, specify the **Precision** of the numbers under the **Alternate units tolerance**.
- Specify the **Zero suppression** for both the **Primary units** tolerance and the **Alternate units** tolerance.

9.11 CREATING A SUB STYLE

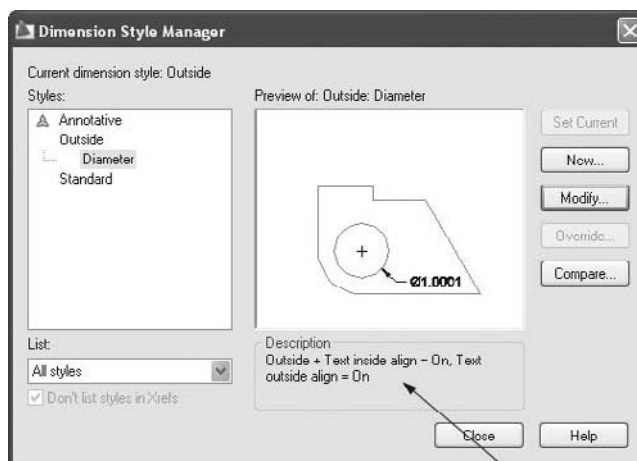
- Sometimes you need a dimension style identical to almost all types of dimensions except for **Diameter**, for example.
- In this case, we create a dimension style for all types, and then we create a sub dimension style from it.
- Perform the following steps:
 - Create your dimension style.
 - Select it from the list in the **Dimension Style** dialog box.
 - Click the **New** button to create a new style.
 - The following dialog box will appear:



- For **Use for**, select **Diameter dimensions** (for example). The dialog box will change to:



- Now, click **Continue** and make the changes you want. These changes only will affect the diameter dimensions.
- In the **Dimension Style** dialog box, you will see something like the following:



The differences between the dimension style and the sub-style

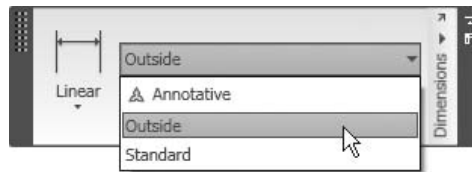
- You can see how AutoCAD relates the new sub style to the existing style. You can also see in the **Description** area how AutoCAD lists the differences between the style and its sub style.

9.12 CONTROLLING DIMENSION STYLES

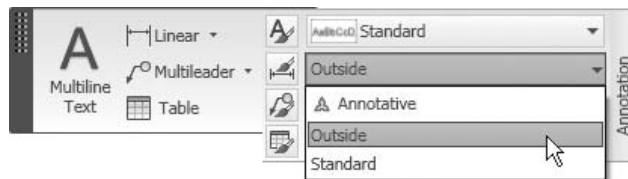
- Once you have created more than one dimension style, you can control these dimension styles using the **Current**, **Modify**, and **Delete** buttons.

Set Dimension Style Current

- While you are in the **Dimension style** dialog box, select the desired dimension style and click the **Current** button.
- Make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, select the current dimension style from the pop-up list:



- Make sure you are in the **Home** tab on the **Ribbon**, and, expanding the **Annotation** panel, use the pop-up list to set the current dimension style:



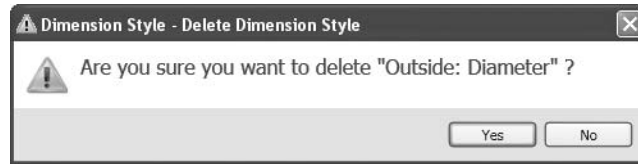
Modify Dimension Style

- While you are in the **Dimension Style** dialog box, select the desired dimension style and click the **Modify** button. The same dialog box will appear for more editing.

Delete Dimension Style

- In order to delete a dimension style there are three conditions:
 - It should be not used in the current drawing.
 - It should not be the current dimension style.
 - It should not have any sub style. If it does, delete the sub style (child) first and then delete the main style (parent).

- If these three conditions are fulfilled, select the desired dimension style to be deleted and press [Del] (you can also select it and right-click, and then select **Delete**). The following dialog box will appear:



- If you click **Yes**, it will be deleted. If you click **No**, the whole process will be canceled.

CREATING DIMENSION STYLES (METRIC)



Workshop 17-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_17.dwg*.
3. Create a new dimension style and name it **Outside**, starting from **Standard**, and use for **All dimensions**. (Anything not mentioned here should be left at the default value or setting.)
4. Under **Line** make the following changes:
 - a. **Extend beyond dim line** = 0.25
 - b. **Offset from origin** = 0.15
5. Under **Symbols and Arrows** make the following changes:
 - a. **Arrowhead, First** = Oblique
 - b. **Arrow size** = 0.25
6. Under **Text** make the following changes:
 - a. **Text style** = Dimension
 - b. **Text placement, Vertical** = Above
 - c. **Text alignment** = Aligned with dimension line
7. Under **Fit** make the following changes:
 - a. **Use overall scale** = 1000
8. Under **Primary Units** make the following changes:
 - a. **Linear dimension, Precision** = 0.00
 - b. **Suffix** = m
 - c. **Scale Factor** = 0.001
9. Create a new style and name it **Inside**, starting from **Outside**, and use for **All dimensions**.

10. Under **Lines** make the following changes:
 - a. **Extension lines, Suppress Ext line 1** = on, **Ext line 2** = on
11. Under **Symbols and Arrows** make the following changes:
 - a. **Arrow size** = 0.20
12. Under **Text** make the following changes:
 - a. **Text style** = Standard
 - b. **Text height** = 0.25
13. Under **Fit** make the following changes:
 - a. **Fine tuning, Place text manually** = on
14. Make a sub style from **Outside** for **Radius dimensions**.
15. Under **Symbols and Arrows** make the following changes:
 - a. **Arrowheads, Second** = Closed filled
16. Under **Text** make the following changes:
 - a. **Text alignment** = ISO Standard
17. Save the file and close it.

CREATING DIMENSION STYLES (IMPERIAL)



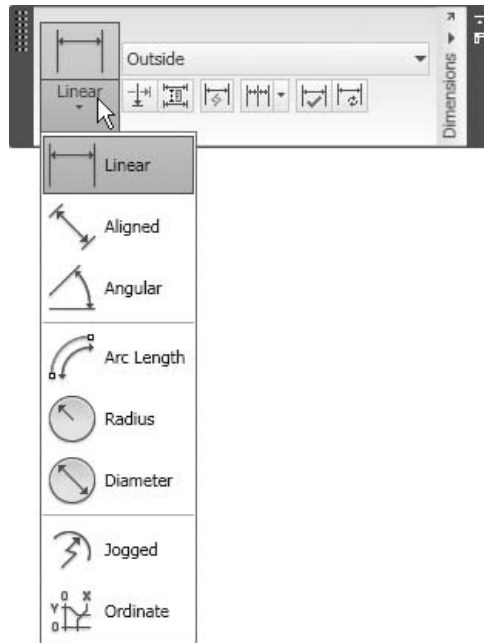
Workshop 17-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_17.dwg*.
3. Create a new dimension style and name it **Outside**, starting from **Standard**, and use for **All dimensions**. (Anything not mentioned here should be left at the default value or setting.)
4. Under **Line** make the following changes:
 - a. **Extend beyond dim line** = 3/4"
 - b. **Offset from origin** = 1/2"
5. Under **Symbols and Arrows** make the following changes:
 - a. **Arrowhead, First** = Oblique
 - b. **Arrow size** = 3/4"
6. Under **Text** make the following changes:
 - a. **Text style** = Dimension
 - b. **Text placement, Vertical** = Above
 - c. **Text alignment** = Aligned with dimension line

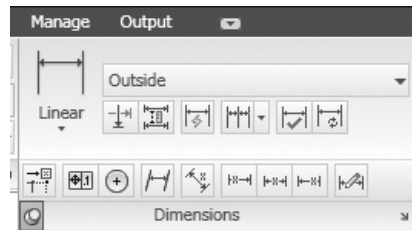
7. Under **Fit** make the following changes:
 - a. **Use overall scale** = 12
8. Under **Primary Units** make the following changes:
 - a. **Unit Format** = Architectural, **Precision** = 0'-0"
 - b. **Suffix** = ft
9. Create a new style and name it **Inside**, starting from **Outside**, and use for **All dimensions**.
10. Under **Lines** make the following changes:
 - a. **Extension lines**, **Suppress Ext line 1** = on, **Ext line 2** = on
11. Under **Text** make the following changes:
 - a. **Text style** = Standard (change the font to be Arial)
 - b. **Text height** = 3/4"
12. Under **Fit** make the following changes:
 - a. **Fine tuning**, **Place text manually** = on
13. Make a sub style from **Outside** for **Radius dimensions**.
14. Under **Symbols and Arrows** make the following changes:
 - a. **Arrowheads**, **Second** = Closed filled
15. Under **Text** make the following changes:
 - a. **Text alignment** = ISO Standard.
16. Save the file and close it.

9.13 AN INTRODUCTION TO DIMENSIONING COMMANDS

- Dimensioning commands insert dimensions using the points specified by a user.
- You will find all commands under the **Annotate** tab in the **Dimensions** panel:



- More commands can be found in the hidden part of the **Dimensions** panel:



- For the purpose of this book, all dimensioning commands will be found in the **Dimensions** panel.

9.14 THE LINEAR COMMAND

- Use the **Linear** command to create a horizontal or vertical dimension.
- To start the **Linear** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Linear** button.
- The following prompt will appear:

Specify first extension line origin or <select object>:

(Specify the first point)

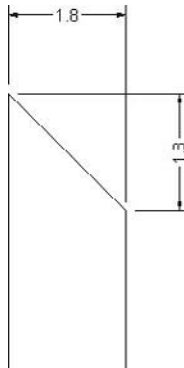
Specify second extension line origin: **(Specify the second point)**

Specify dimension line location or

[Mtext/Text/Angle/Horizontal/Vertical/Rotated]:

(Specify the location of the dimension line)

- There are three steps to follow:
 - Specify the first point of the dimension distance to be measured.
 - Specify the second point of the dimension distance to be measured.
 - Specify the location of the dimension block by specifying the location of the dimension line.
- The following is the result:



- You can use the other options available. They are:
 - **Mtext**
 - **Text**
 - **Angle**
 - **Horizontal**
 - **Vertical**
 - **Rotated**

Mtext

- To edit the measured distance in the **MTEXT** command.

Text

- To edit the measured distance in the **DTEXT** command.

Angle

- To change the angle of the text.

Horizontal

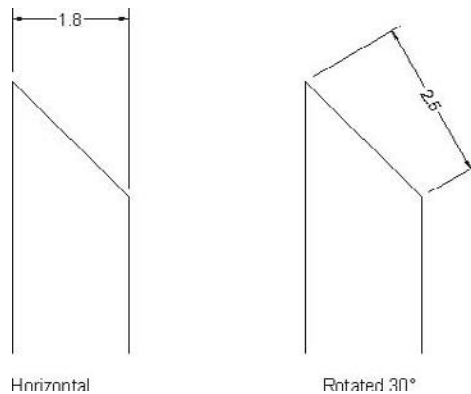
- To create a horizontal dimension.

Vertical

- To create a vertical dimension.

Rotated

- To create a dimension line parallel to another angle given by the user. As in the case of projecting a distance over another angle. See the following illustration:



9.15 THE ALIGNED COMMAND

- The **Aligned** command is used to create a dimension parallel to the measured distance.
- To start the **Aligned** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Aligned** button.
- The following prompt will appear:

Specify first extension line origin or <select object>:

(Specify the first point)

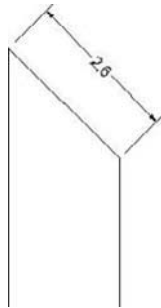
Specify second extension line origin: *(Specify the second point)*

Specify dimension line location or

[Mtext/Text/Angle]: *(Specify the location of the dimension line)*

- There are three steps to follow:
 - Specify the first point of the dimension distance to be measured.

- Specify the second point of the dimension distance to be measured.
 - Specify the location of the dimension block by specifying the location of the dimension line.
- The following is the result:



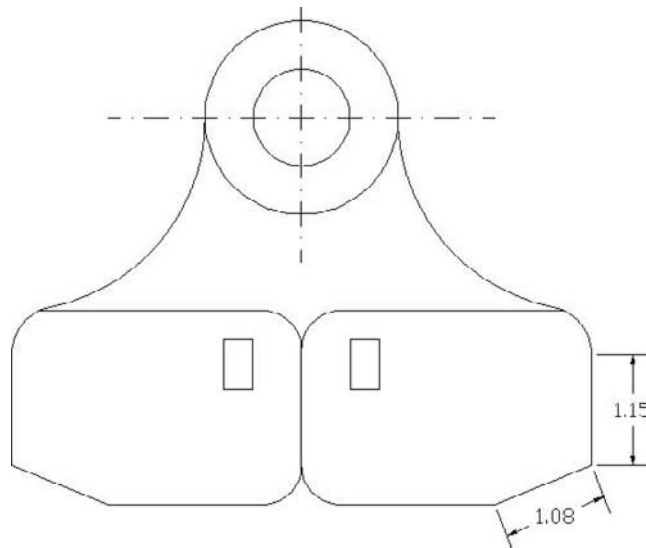
- The rest of the options are the same as the **Linear** command.

LINEAR AND ALIGNED DIMENSIONS



Exercise 34

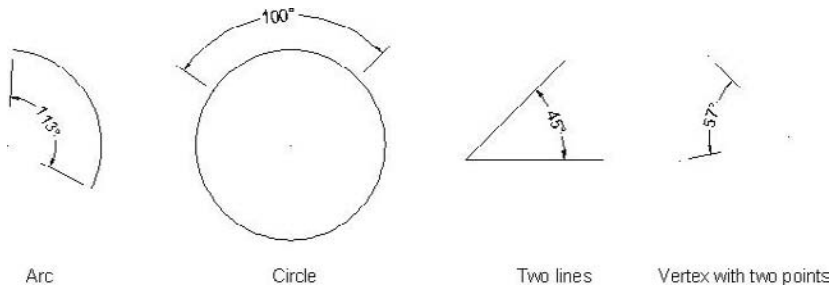
1. Start AutoCAD 2010.
2. Open the file *Exercise_34.dwg*.
3. Make the **Dimension** layer current.
4. Make the linear and aligned dimensions as shown:



5. Save the file and close it.

9.16 THE ANGULAR COMMAND

- The **Angular** command is used to create an angular dimension.
- To start the **Angular** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Angular** button.
- There are four ways to place an angular dimension in AutoCAD:
 - Select an arc and AutoCAD will measure the included angle.
 - Select a circle. The position that you select the circle from will be the first point and the center of the circle will be the second point. AutoCAD will ask the user to specify any point on the diameter of the circle and will place the angle accordingly.
 - Select two lines. AutoCAD will measure either the inside angle or the outside angle.
 - Select a vertex, which will be considered a center point, then AutoCAD will ask you to specify two points and will measure either the inside angle or the outside angle.



- The following prompts will appear:

Select arc, circle, line, or <specify vertex>: (*Select the desired method as discussed above—assume we select arc*)
 Specify dimension arc line location or [Mtext/Text/Angle]:
 (*Specify the dimension block location*)

9.17 THE ARC LENGTH COMMAND

- The **Arc Length** command is used to create a dimension showing the length of the selected arc.
- To start the **Arc Length** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Arc Length** button.
- The following prompt will appear:

Select arc or polyline arc segment: *(Select the desired arc)*
Specify arc length dimension location, or [Mtext/Text/Angle/
Partial/Leader]: *(Specify the location of the dimension block)*

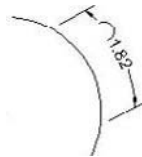
- There are two steps to follow:
 - Select the desired arc.
 - Specify the location of the dimension block.
- The following is the result:



- The options **Mtext**, **Text**, and **Angle**—which are available to the **Arc Length** command—were discussed in the **Linear** command section.

Partial

- If you want **Arc Length** to measure part of the arc and not the entire arc, specify the two points on the arc. The result looks something like this:



Leader

- To add a leader, see the following example:



9.18 THE RADIUS COMMAND

- The **Radius** command is used to put a radius dimension on an arc and/or a circle.
- To start the **Radius** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Radius** button.

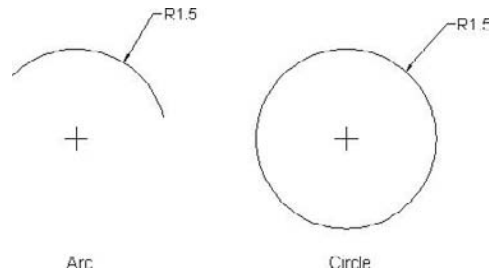
- The following prompts will appear:

Select arc or circle: (*Select the desired arc or circle*)

Specify dimension line location, or

[Mtext/Text/Angle]: (*Specify the location of the dimension block*)

- There are two steps to follow:
 - Select the desired arc or circle.
 - Specify the location of the dimension block.
- As a result, you will see:



9.19 THE DIAMETER COMMAND

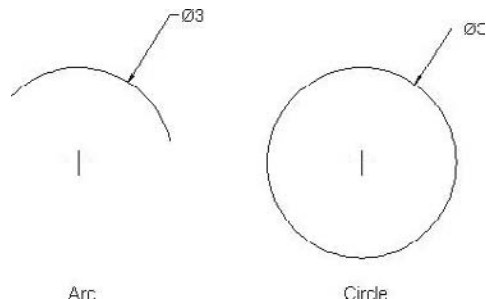
- The **Diameter** command is used to put diameter dimensions on an arc and/or a circle.
- To start the **Diameter** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Diameter** button.
- The following prompts will appear:

Select arc or circle: (*Select the desired arc or circle*)

Specify dimension line location, or

[Mtext/Text/Angle]: (*Specify the dimension block location*)

- As a result, you will see the following:



9.20 THE JOGGED COMMAND

- If you have a large arc, then its center is very far from the arc itself. Therefore, it is difficult for users to add a normal radius dimension.
- The solution is to add a jogged radius dimension.
- The **Jogged** command is used to add a jogged radius to an arc or a circle.
- To start the **Jogged** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Jogged** button.
- The following prompts will appear:

Select arc or circle: *(Select the desired arc or circle)*

Specify center location override: *(Specify a point which will act as a new center point)*

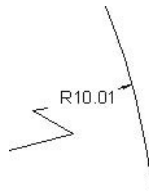
Dimension text = 1.5

Specify dimension line location or

[Mtext/Text/Angle]: *(Specify the dimension line location)*

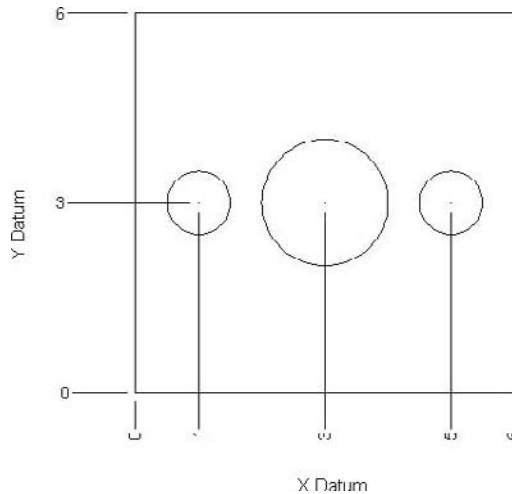
Specify jog location: *(Specify the location of the jog)*

- There are four steps to follow:
 - Select the desired arc or circle.
 - Specify the point that will act as the new center point.
 - Specify the dimension line location.
 - Specify the location of the jog.
- Once you complete the four steps, you will see:



9.21 THE ORDINATE COMMAND

- The **Ordinate** command is used to add several measurements to objects relative to a certain point.
- To start the **Ordinate** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Ordinate** button.
- The **Ordinate** command allows you to set dimensions relative to a datum, either in X or in Y. See the following illustration:



- You have to change the UCS (User Coordinate System) origin location to the desired location, otherwise, the values will be relative to the current location of 0,0.
- The following prompts will appear:

Specify feature location: (*Click on the desired point*)
 Specify leader endpoint or
 [Xdatum/Ydatum/Mtext/Text/Angle]: (*Specify the dimension location*)

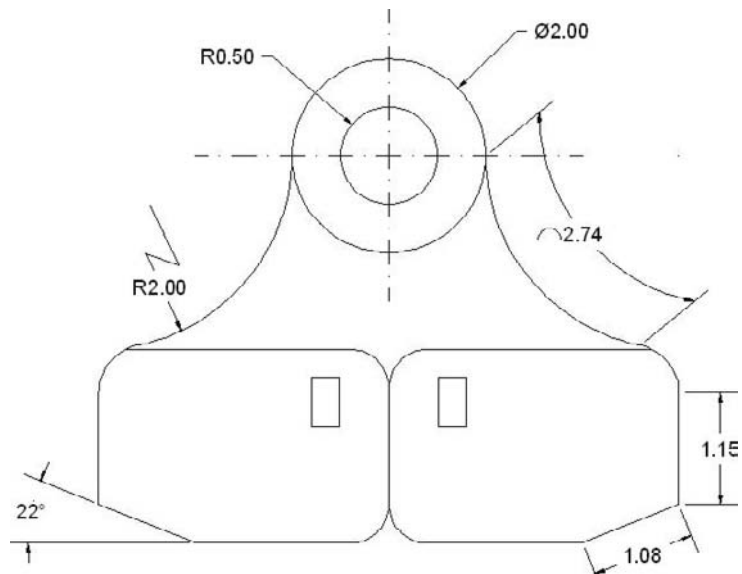
- By default, when you select a point, you may go in the direction of X or Y. If you want the **Ordinate** command to go exclusively in the X direction, then select the **Xdatum** option, or select **Ydatum** if you want to go exclusively in the Y direction.
- The additional options are the same as those in the **Linear** command.

ANGULAR, ARC LENGTH, RADIUS, DIAMETER, AND DIMENSIONS



Exercise 35

1. Start AutoCAD 2010.
2. Open the file *Exercise_35.dwg*.
3. Make the following modification to the current dimension style (i.e., **Standard**):
 - a. Under **Text**, change the current text style to use **Arial** font.
4. Perform the steps for the five types of dimensions as shown:



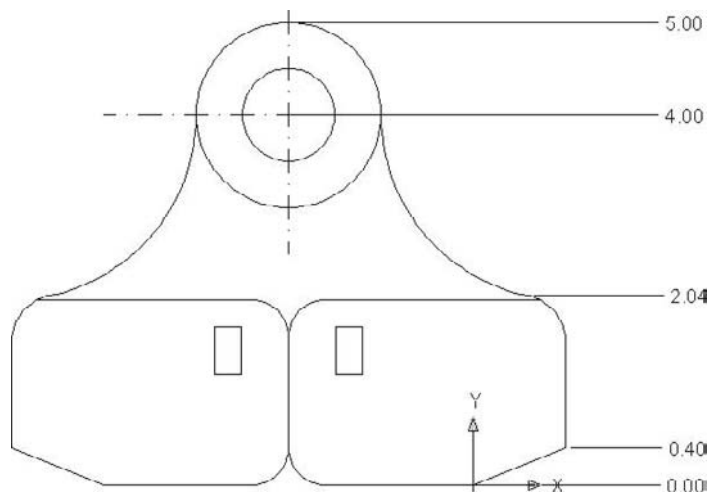
5. Save the file and close it.

ORDINATE AND JOGGED



Exercise 36

1. Start AutoCAD 2010.
2. Open the file *Exercise_36.dwg*.
3. Create the ordinate and jogged dimensions as shown:



4. Save the file and close it.

9.22 THE CONTINUE COMMAND

- After you put a dimension in your drawing (i.e., linear, aligned, angular, or ordinate), you can ask AutoCAD to continue using the same type and allocate it along the first one.
- The **Continue** command will allow you to input many dimensions swiftly.
- To start the **Continue** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Continue** button.

If No Dimension Was Created in This Session

- The following prompt will appear:

Select continued dimension: (*Select either Linear, Aligned, Ordinate, or Angular*)

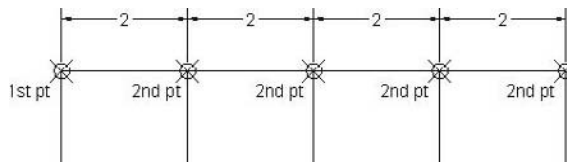
- AutoCAD will consider the selected dimension as the base dimension and will continue accordingly.

If a Dimension Was Created in This Session

- The following prompt will appear:

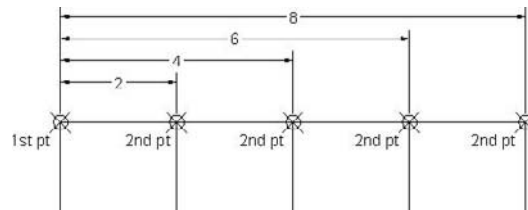
Specify a second extension line origin or [Undo/Select]
<Select>: (*Specify the second point of the last Linear, Aligned, Ordinate, or Angular, or select an existing dimension*)

- AutoCAD will give you the ability to do one of three things:
 - If you already input a linear dimension (for example), then you can continue by specifying the second point, considering the second point of the first dimension is the first point of the continuing dimension.
 - You can select an existing dimension and continue from there.
 - You can undo the last continue dimension.
- See the following illustration:



9.23 THE BASELINE COMMAND

- The **Baseline** command works just like the **Continue** command except the dimensions will always be related to the first point the user selected.
- To start the **Baseline** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Baseline** button.
- All of the prompts and procedures are identical to the **Continue** command.
- See the following illustration:

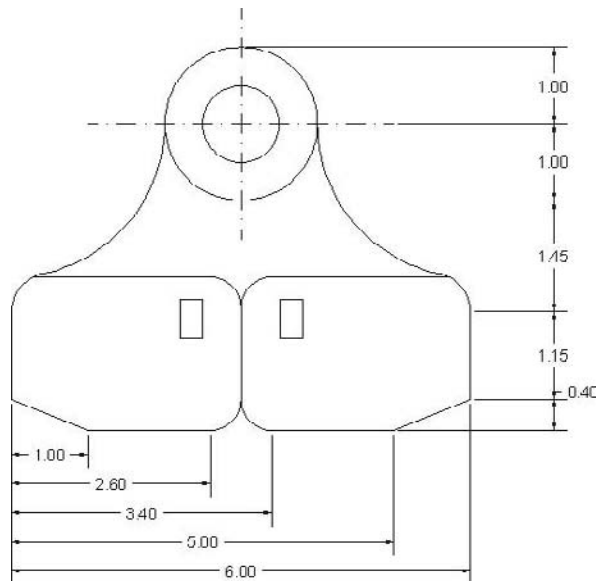


CONTINUOUS AND BASELINE DIMENSIONS



Exercise 37

1. Start AutoCAD 2010.
2. Open the file *Exercise_37.dwg*.
3. Create the continuous and baseline dimensions as shown:



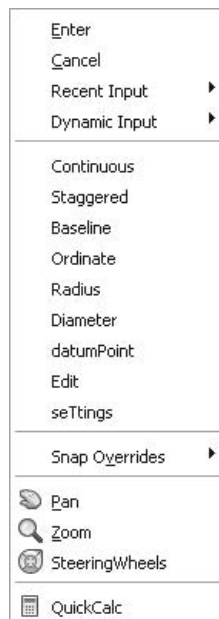
4. Save the file and close it.

9.24 THE QUICK DIMENSION COMMAND

- The **Quick Dimension** command is used to place a group of dimensions in a single step.
- To start the **Quick Dimension** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Dimensions** panel, click the **Quick Dimension** button.
- The following prompt will appear:

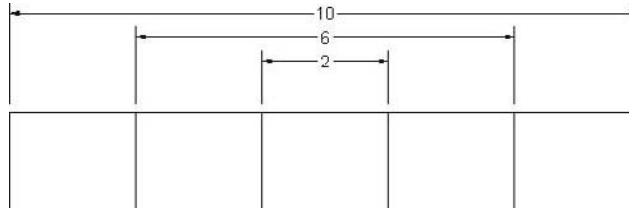
Select geometry to dimension: (*Either by clicking, Window, or Crossing, after you are done press [Enter]*)
Specify dimension line position, or
[Continuous/Staggered/Baseline/Ordinate/Radius/
Diameter/datumPoint/Edit/seTtings] <Continuous>:

- At this prompt you can right-click and the following shortcut menu will appear:



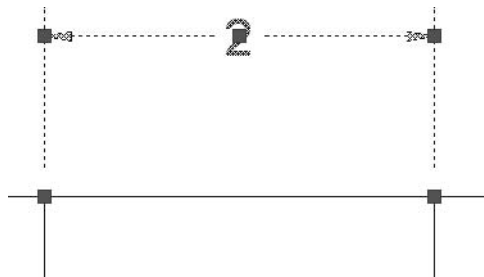
- From this shortcut menu, you can select the proper dimension type. You can choose from: **Continuous**, **Staggered**, **Baseline**, **Ordinate**, **Radius**, or **Diameter**.
- Select the type and then specify the dimension line location; a group of dimensions will be placed in a single step.

- See the following **Staggered** example:

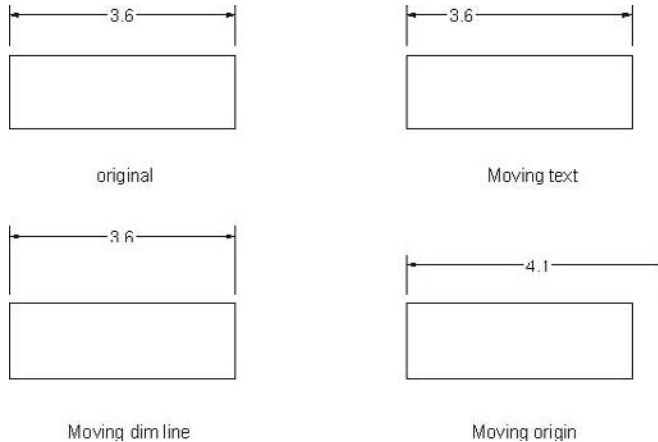


9.25 DIMENSION BLOCKS AND GRIPS

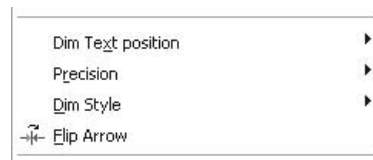
- You can edit dimension blocks using **Grips**.
- If you click a dimension block, five grips will appear, just like the following:



- As you can see, the grips appear in the following places:
 - The two ends of the dimension line.
 - The two origins of the dimension line.
 - The dimension text.
- You can change the position of the text by clicking its grip and moving it parallel to the dimension line.
- You can change the position of the dimension line by clicking one of the two grips and moving it closer to, or farther from, the origin.
- You can change the measured distance by moving one of the two grips of the origin, so the distance will change accordingly.
- See the following example:



- Also, if you select a dimension block and right-click, the following shortcut menu will appear (only the part concerning the dimension is shown):



- You can change **Dim Text Position**, **Precision**, **Dim Style**, and **Flip Arrow** in the selected dimension block.

Dim Text Position

- You can change the position of the dimension text as seen in the example.

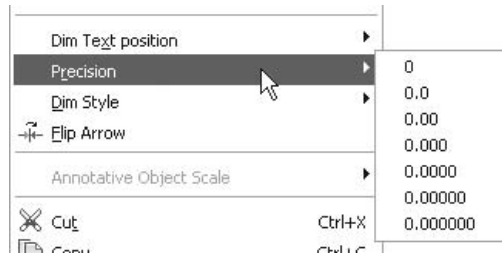


- **Above dim line:** The dimension text will move from any other position to **Above**.
- **Centered:** The dimension text will move from any other position to **Centered**.
- **Home text:** The dimension text will be restored to its original position according to its dimension style.

- **Move text alone:** Allows you to position the text freely.
- **Move with leader:** Allows you to position the text freely, but with a leader.
- **Move with dim line:** Allows you to move both the dimension text and the dimension line in one single step.

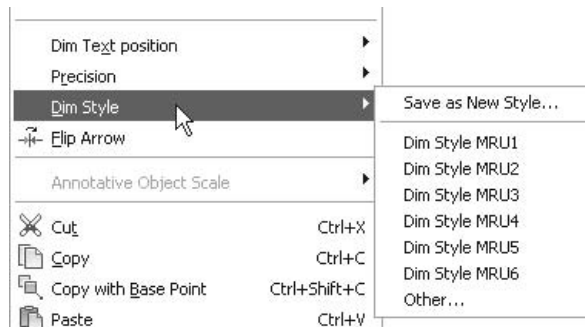
Precision

- This will allow users to set the number of decimal places for the number shown.
- You can start from zero decimal places up to eight decimal places.



Dim Style

- The changes made using this method can be saved in a new dimension style using the **Save as New Style** option. Type in a new name.
- You can change the dimension style of any dimension block in the drawing. The existing dimension styles will appear in a pop-up menu; select the new desired dimension style.

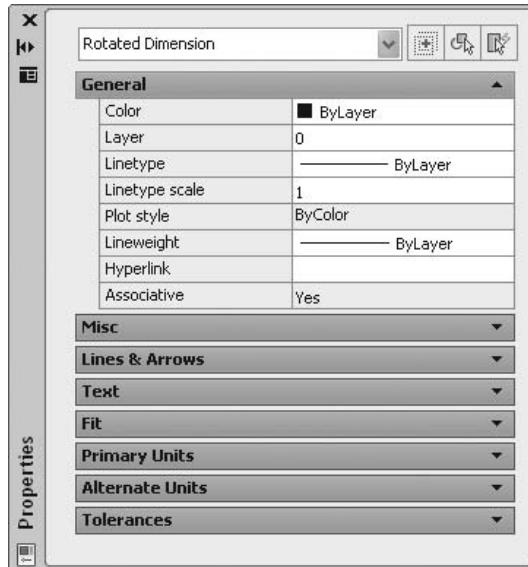


Flip Arrow

- This method is used to flip the arrows from inside to outside, and vice versa.
- The process can be done on only one arrow at a time.

9.26 DIMENSION BLOCK PROPERTIES

- Select a dimension block and right-click, and then select **Properties**. The following dialog box will appear:



- Under **General**, you will see the general properties of the selected dimension block.
- You will see the following properties: **Misc**, **Lines & Arrows**, **Text**, **Fit**, **Primary Units**, **Alternate Units**, and **Tolerances**. If you compare these to the **Dimension Style** command, you will find them identical, which means you can change any of the characteristics of the dimension block after it goes through **Properties**.

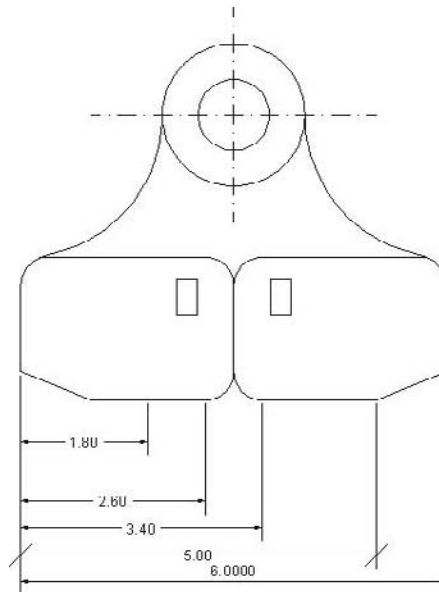
QUICK DIMENSION AND EDITING



Exercise 38

1. Start AutoCAD 2010.
2. Open the file *Exercise_38.dwg*.
3. Using the **Quick Dimension** command and **Crossing**, select all of the lines and the two arcs at the bottom of the shape.
4. Deselect the two arcs along with the middle vertical line and press [Enter], then right-click and select the baseline, and put it in a convenient place beneath the shape.

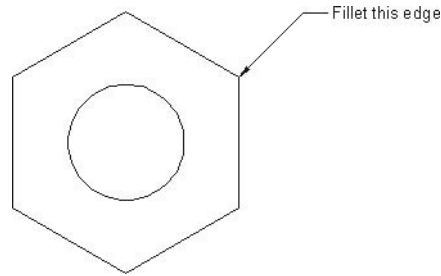
5. Select the dimension with **value** = 1.00 (the left-most spot). Five grips will appear.
6. Make sure that **Middle** in **OSNAP** is turned on. Select the upper right-hand grip to make it hot and move it to the middle point of the right horizontal line.
7. The value of the dimension should read 1.80. Press [Esc] twice.
8. Select the same dimension block again and move the dimension line closer to the shape. Press [Esc] twice.
9. Select the lower dimension with **value** = 6.00.
10. Right-click and change the **Precision** of the number to 0.0000 and make it **Above dim line**.
11. Select the dimension with **value** = 5.00 and show the **Properties** palette.
12. Change **Arrow 1** and **Arrow 2** to be **Oblique**, and the **arrow size** = 0.3.
13. The shape should look like the following:



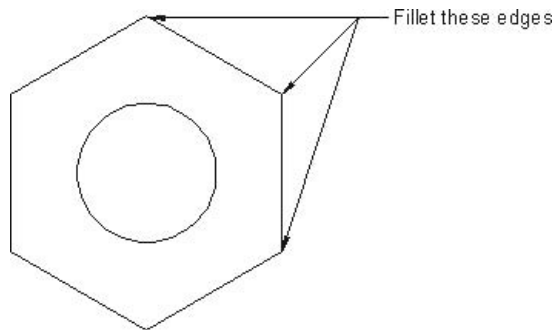
14. Save the file and close it.

9.27 AN INTRODUCTION TO THE MULTILEADER

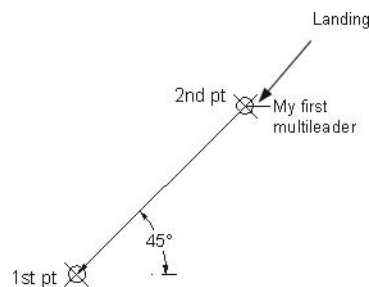
- A leader in AutoCAD is an arrow pointing to a part of the drawing, with two lines and some text to explain certain facts about that part.
- A multileader can be one arrow with a single set of lines or multiple arrows with multiple sets of lines. See the following illustration of a single leader:



- See the following illustration of a multileader:



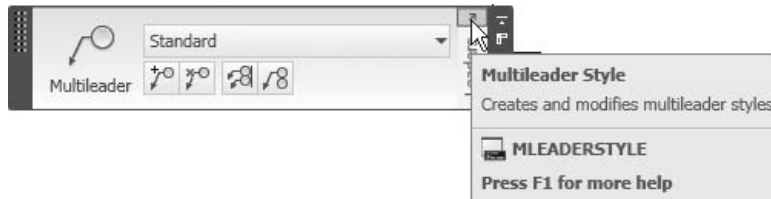
- Normally, you will create a single leader and you can create a multileader from it using other commands.
- You will specify (by default) two points; the first one is where the arrow will be pointing to and the second one will specify the length and angle of the leader. Another small horizontal landing will be added automatically. See the following illustration:



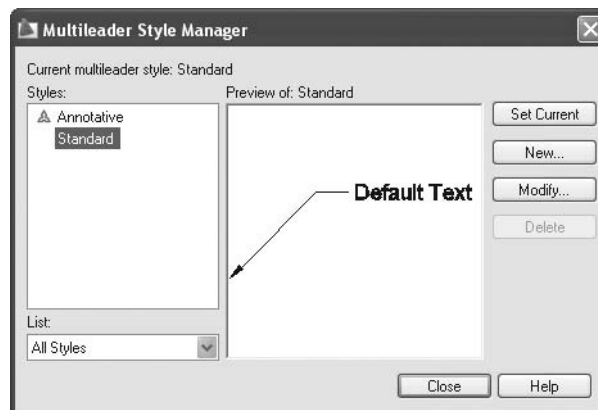
- A multileader has its own style and set of commands, which we will discuss in the coming pages.

9.28 MULTILEADER: CREATING THE STYLE

- As we discussed in **Dimension Style**, you will use **Multileader Style** to set the characteristics of the multileader block.
- To start the **Multileader Style** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Leaders** panel, click the **Multileader Style** button.



- The following dialog box will appear:



- To create a new style, click the **New** button. The following dialog box will appear:

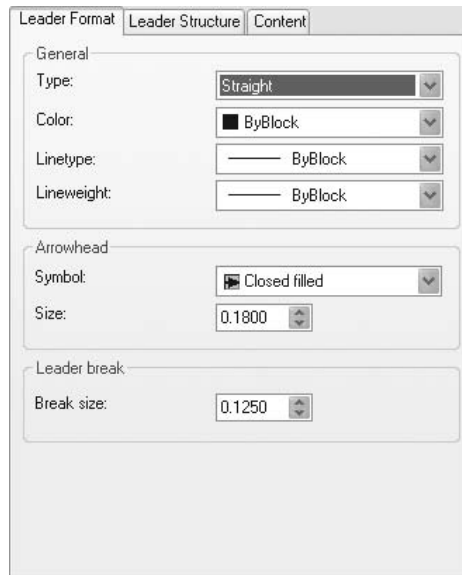


- Type the name of the new style and select the existing style that you want to start with. Click **Continue** to start modifying the **Multileader style**.

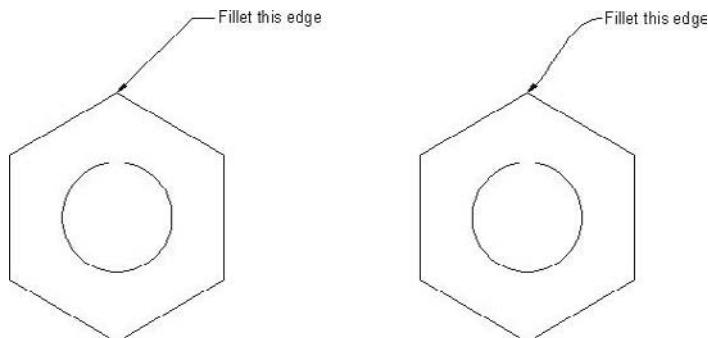
- You will see three tabs:
 - **Leader Format**
 - **Leader Structure**
 - **Content**
- We will discuss each one of these in the coming pages.

Leader Format

- You will see the following dialog box:



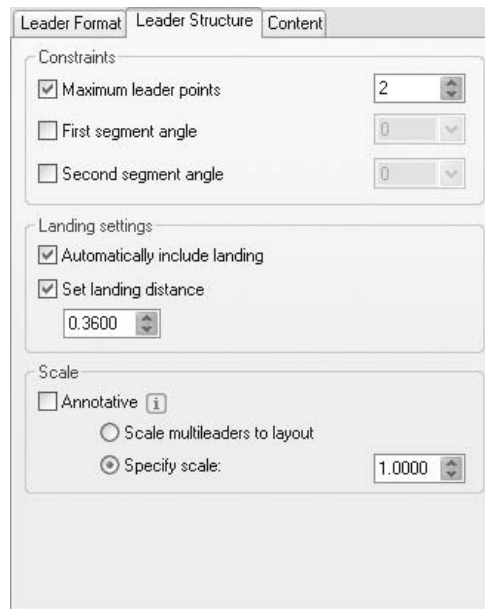
- You have three choices for the type of leader:
 - **Straight**
 - **Spline**
 - **None**
- See the following illustration:



- You can choose the **Color**, **Linetype**, and **Lineweight** in the same way as the **Dimension Style** command.
- Choose the **Arrowhead Symbol** (from 20 different existing shapes) and its size.
- If there is **Dimension Break**, how large will the break of the multileader be? Answer this question by setting the **Break size** value.

Leader Structure

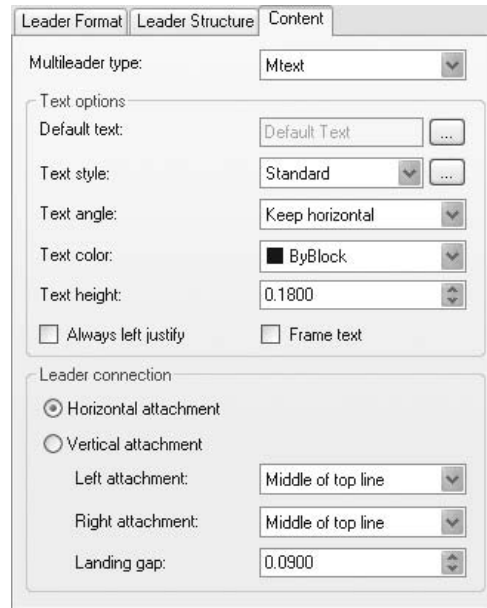
- You will see the following dialog box:



- Specify the **Maximum leader points**. As discussed earlier, the default value is 2, and you can specify more points.
- By default, you will specify the **First segment angle** and a small horizontal landing (**Second segment angle**) will be added, but you can set the angle values for both in the **Multileader style**.
- Choose whether to **Automatically include landing** or not. If you select to include it, you must determine the **landing length**.
- Select whether the multileader will be **Annotative** or not. (We will leave it off for now.)

Content

- You will see the following dialog box:

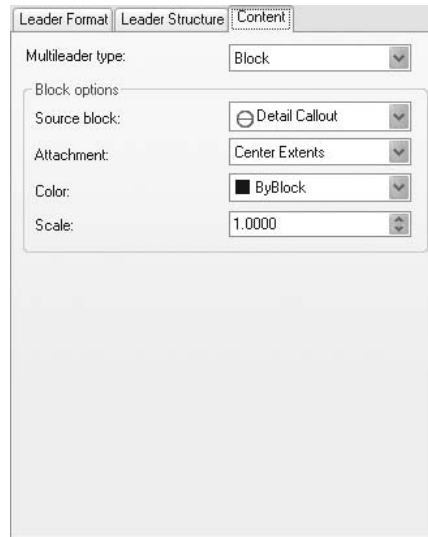


- There are two types of **Multileader**:
 - With **Mtext**
 - With a **Block** (either predefined or user-defined)
- See the following illustration:



- **Mtext** (Multiline text) is selected by default. To adjust the settings:
 - If there is any text that should appear each time, type it in **Default text**.
 - Specify the **Text style**.
 - Specify the **Text angle** (**Keep horizontal**, **Right-reading**, or **As inserted**).
 - Specify the **Text Color** and **Text height** (if **Text Style** height = 0).
 - Specify whether to make the text **Always left justify** or not, and with **Frame** or not.
 - Control the position of the text relative to the landing for both left and right leader lines.
 - Control the gap distance between the end of the landing and the text.

- If you select the **Multileader** type to be **Block**, you will see the following dialog box:



- Adjust the following settings:
 - Specify the **Source block** (choose from the list) or select **User Block**. The following dialog box will appear:

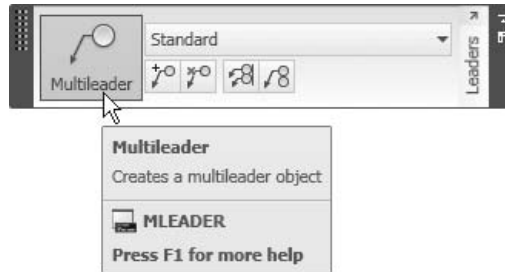


- Type the name of the desired block and click **OK**.
- Specify the **Attachment** position.
- Specify the **Color** of the attachment.
- Specify the **Scale** of the attachment.

9.29 MULTILEADER COMMANDS

- After you finish creating **Multileader Style**, you are now ready to insert the multileader into your drawing.
- You will always start with the **Multileader** command to insert a single leader.

- To start the **Multileader** command, make sure you are in the **Annotate** tab on the **Ribbon**, and, using the **Leaders** panel, click the **Multileader** button.



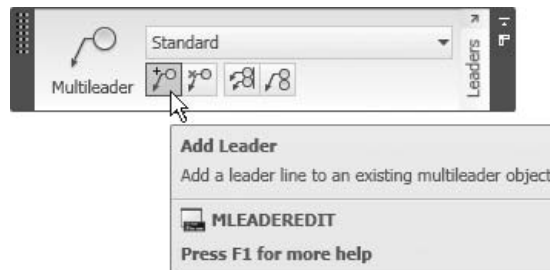
- The following prompts will appear:

Specify leader arrowhead location or [leader Landing first/
Content first/Options] <Options>:

(Specify the point which the arrow should point to)

Specify leader landing location: *(Specify the angle and the
length of the leader)*

- Type in the text you want to appear in the leader.
- Click the **Add Leader** button to add more leaders (arrows) to an existing single leader:



- The following prompts will appear:

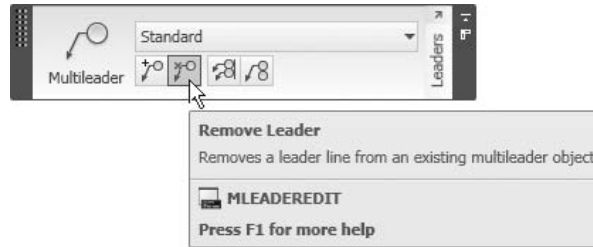
Select a multileader:

1 found

Specify leader arrowhead location:

Specify leader arrowhead location:

- Click the **Remove Leader** button to remove some leaders from an existing multileader.



- The following prompts will appear:

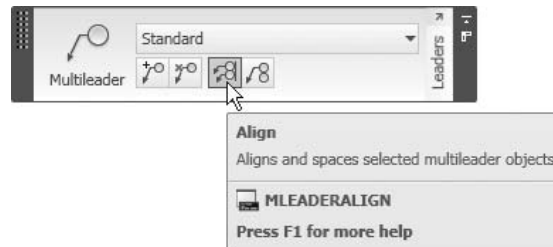
Select a multileader:

1 found

Specify leaders to remove:

Specify leaders to remove:

- Click the **Align** button to arrange several multileaders in the same line.



- The following prompts will appear:

Select multileaders: 1 found

Select multileaders: 1 found, 2 total

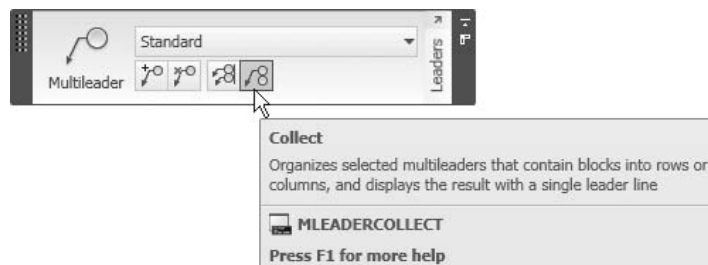
Select multileaders: (**Press [Enter]**)

Current mode: Use current spacing

Select multileader to align to or [Options]:

Specify direction:

- Click the **Collect** button to collect several multileaders into a single leader. This command only works with leaders containing blocks:



- You will see the following prompts:

Select multileaders:

Select multileaders: (**Press [Enter] when you are done**)

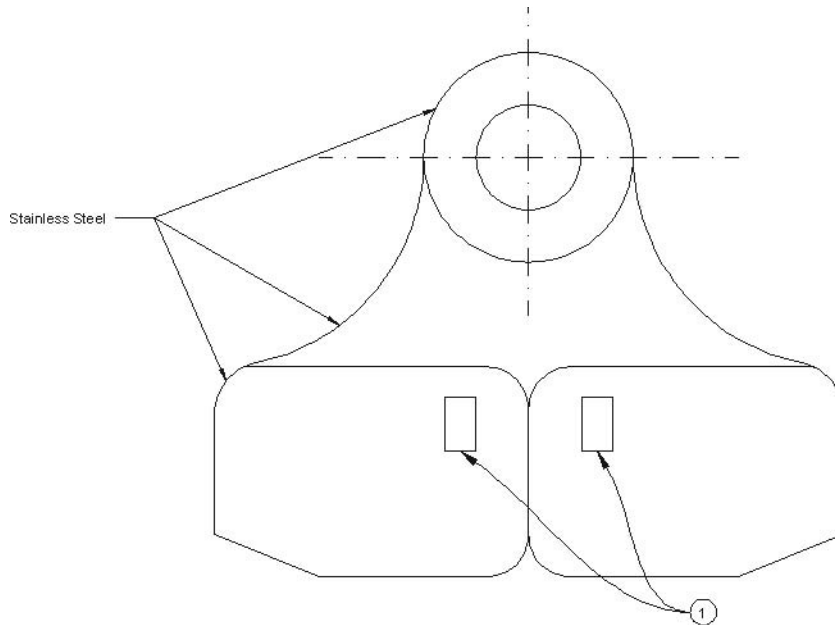
Specify collected multileader location or [Vertical/
Horizontal/Wrap] <Horizontal>:

MULTILEADER



Exercise 39

1. Start AutoCAD 2010.
2. Open the file *Exercise_39.dwg*.
3. Using **Material style** and **Free-Block style**, add the following multileader:



4. Save the file and close it.

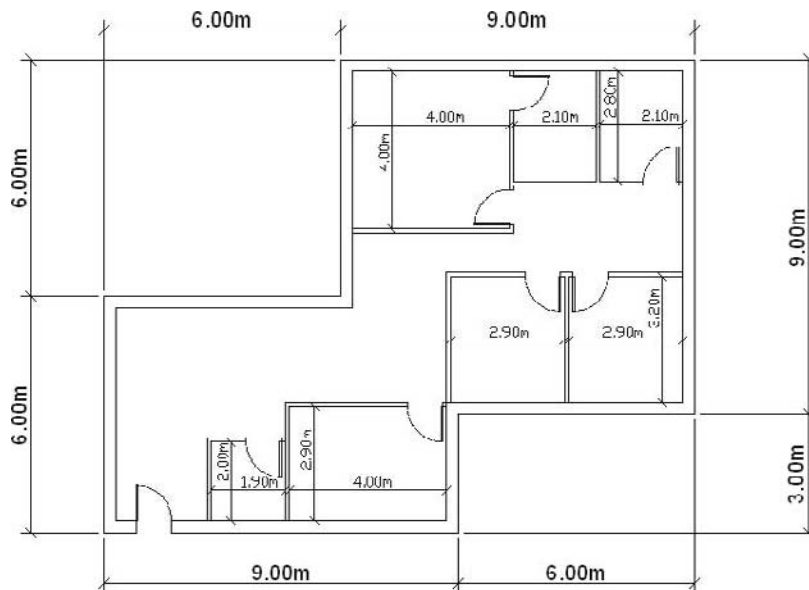
PUTTING DIMENSIONS ON THE PLAN (METRIC)



Workshop 18-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_18.dwg*.

3. Make the **Dimension** layer the current layer.
4. Freeze the following layers: **Furniture**, **Hatch**, and **Text**.
5. Using **Outside** and **Inside** dimension styles, add the dimensions for the outer and inner dimensions as shown:



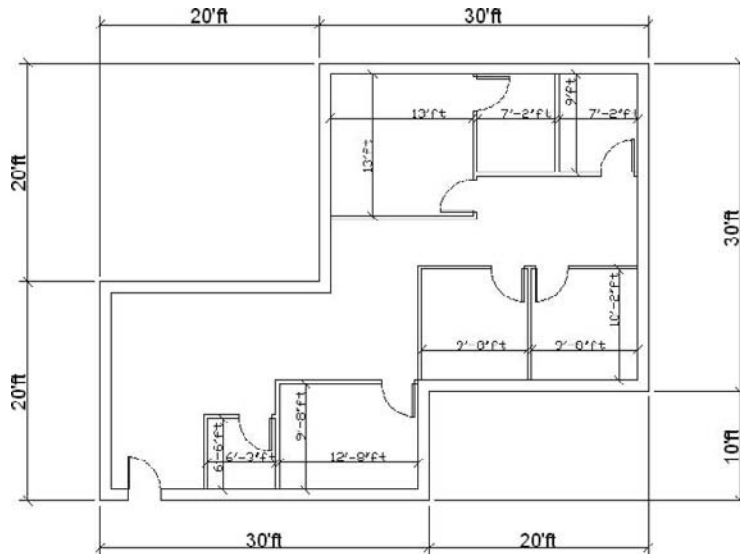
6. Save the file and close it.

PUTTING DIMENSIONS ON THE PLAN (IMPERIAL)



Workshop 18-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_18.dwg*.
3. Make the **Dimension** layer the current layer.
4. Freeze the following layers: **Furniture**, **Hatch**, and **Text**.
5. Using **Outside** and **Inside** dimension styles, add the dimensions for the outer and inner dimensions as shown:



6. Save the file and close it.

CHAPTER REVIEW

1. You can only create dimension styles that will affect all dimension types.
 - a. True
 - b. False
2. _____ and _____ are two types of dimensions that you can use with arcs.
3. Which one of the following is not an AutoCAD dimension command?
 - a. **dimlinear**
 - b. **dimarc**
 - c. **dimchordlength**
 - d. **dimaligned**
4. The only way to add an **Angular dimension** is to have two lines.
 - a. True
 - b. False
5. Which of the following is a type of **Tolerance** in AutoCAD?
 - a. **Deviation**
 - b. **Symmetrical**
 - c. **Limits**
 - d. All of the above

6. In order to make a dimension style _____, double-click the name in the **Dimension Style** dialog box.

CHAPTER REVIEW ANSWERS

1. b
2. **Arc length, Jogged, Radius, Diameter**
3. c
4. b
5. d
6. current

Chapter 10 PLOTTING YOUR DRAWING

In This Chapter

- ◇ Introduction
- ◇ **Model Space** Versus **Paper Space**
- ◇ Introduction to **Layouts**
- ◇ How to Switch between **Model Space** and **Layouts**
- ◇ How to Create a New Layout
- ◇ What Is the **Page Setup Manager**?
- ◇ **Layouts** and **Viewports**
- ◇ Adding **Viewports** to **Layouts**
- ◇ **Model Space** and **Paper Space** Modes in **Layouts**
- ◇ Modifying, Scaling, and Maximizing **Viewports**
- ◇ Freezing Layers in **Viewports**
- ◇ Layer Override in **Viewports**
- ◇ An Introduction to **Plot Style Tables**
- ◇ The **Color-Dependent Plot Style Table**
- ◇ The **Named Plot Style Table**
- ◇ The **Plot** Command
- ◇ What Is a DWF File?
- ◇ What Is a DWFx File?
- ◇ Exporting DWF, DWFx, and PDF Files
- ◇ The **Publish** Command
- ◇ How to View DWF and DWFx Files

10.1 INTRODUCTION

- Before AutoCAD® 2000, almost all AutoCAD users plotted from **Model Space**, which is where they created their design.
- In AutoCAD 2000, the development of **Layouts** made it easy for everyone to shift their attention to the new method, which encompassed many new features and surpassed plotting from **Model Space**.

- Also, in AutoCAD 2000, a new feature was introduced called **Plot Style**, which allowed users to create color-independent configuration plotting.
- Eventhough AutoCAD 2000 was a flagship version in more ways than one, the most important features were the new improvements in the plotting process.

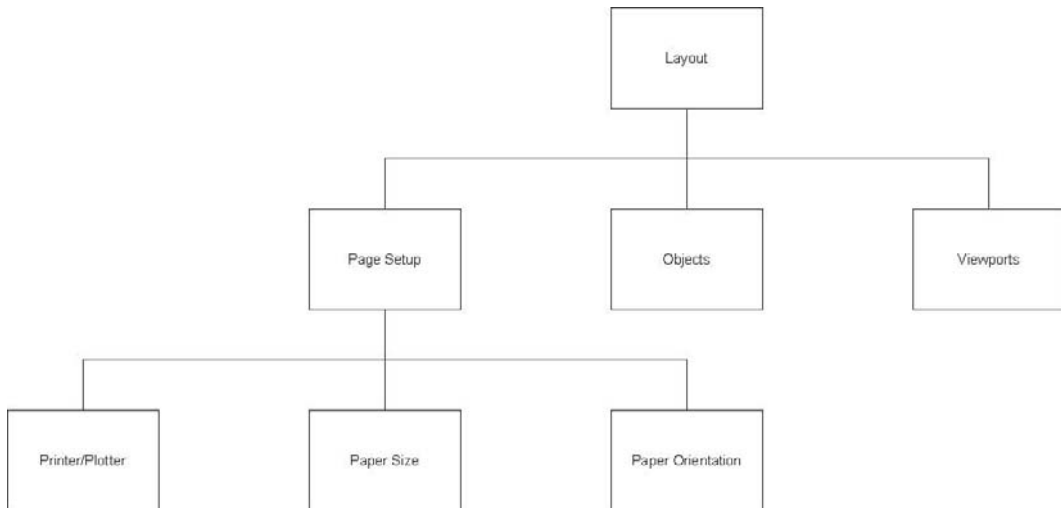
10.2 MODEL SPACE VERSUS PAPER SPACE

- **Model Space** is where you create the drawing using all of the modification processes.
- When you think about plotting, you should use **Paper Space**.
- There is only one **Model Space** in each drawing file.
- Before AutoCAD 2000, there was only one **Paper Space** per drawing file.
- Beginning with AutoCAD 2000, **Paper Space** was changed to **Layout**.
- You can create as many layouts as you wish in each drawing file.
- Each **Layout** will be connected to **Page Setup**, where you will specify at least three things. They are:
 - The plotter to be used.
 - The paper size to be used.
 - The paper orientation (portrait or landscape).
- To demonstrate the importance of this feature, let's take a look at the following example: we have a company who owns an A0 plotter, an A2 printer, and an A4 laser printer. The staff will use all of these printers to print a single drawing.
 - If you use **Model Space**, you will change the setup of the printer, paper size, and paper orientation, along with the drawing area to be plotted each and every time you want to print.
 - But if you create three layouts with the proper **Page Setup**, printing will be fast and easy; simply go to the layout and issue the **Plot** command, which will save time, effort, and money!

10.3 AN INTRODUCTION TO LAYOUTS

- Each layout consists of the **Page Setup**, **Objects**, and **Viewports**.
 - **Page Setup**: Is where you will specify the printer (or plotter), the paper size, and the paper orientation, in addition to other items that will be covered later in this chapter.
 - **Objects**: These include blocks (e.g., the title block), text, dimensions, and any other desired objects.
 - **Viewports**: Viewports will be covered later in this chapter.

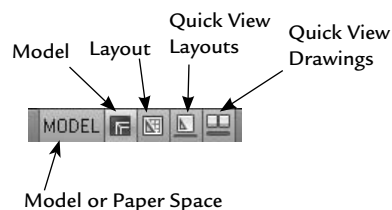
- See the following illustration:



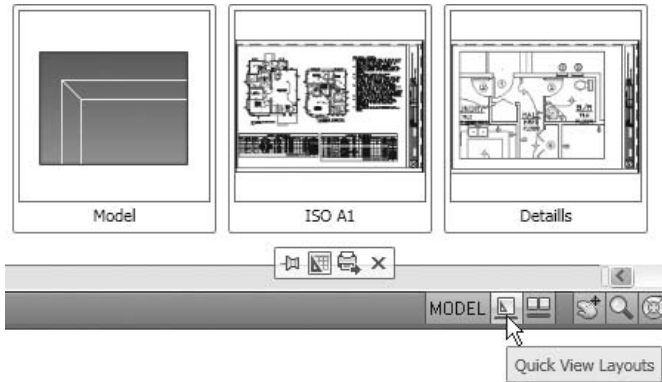
- Each layout should have a name. AutoCAD will give it a temporary name, but you can change it.
- By default, when you create a new drawing using the *acad.dwt* template, two layouts—**Layout1** and **Layout2**—will be created automatically.
- You can choose to have the **Page Setup Manager** dialog box appear when you click on a layout for the first time, for the purpose of setting the printer, paper size, etc.
- **Layouts** and **Page Setup** will be saved in the drawing file.
- By default, **Page Setup** has no name, but you can name it accordingly, use it in other layouts in the current drawing file, or in other drawings.

10.4 HOW TO SWITCH BETWEEN MODEL SPACE AND LAYOUTS

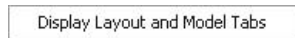
- By default, AutoCAD will be in **Model Space** when you begin a new drawing.
- On the **Status Bar** you will find the following:



- Before you click on the **Layout** button, it is preferable to click the **Quick View Layouts** button to see the available layouts saved in your drawing. See the following illustration:



- In this illustration, you can see two layouts. You can choose which to view by clicking on the small view. Also, you can **Publish**, create a **New Layout**, or choose to **Pin Quick View Layouts**.
- In order to jump from **Model Space** to **Layout** and vice versa, click on the **Model Space** and **Layout** buttons in the **Status Bar**.
- You can also right-click either **Model Space** or **Layout** and the following shortcut menu will appear:



- If you select the **Display Layout and Model Tabs**, you will see the following tabs in the lower left-hand corner of the **Graphical Area**:
- 
- The illustration shows a horizontal row of three tabs: 'Model', 'ISO A1', and 'Details'. The 'Model' tab is currently selected and highlighted.
- This information is essential to what we will discuss in the next section.

10.5 HOW TO CREATE A NEW LAYOUT

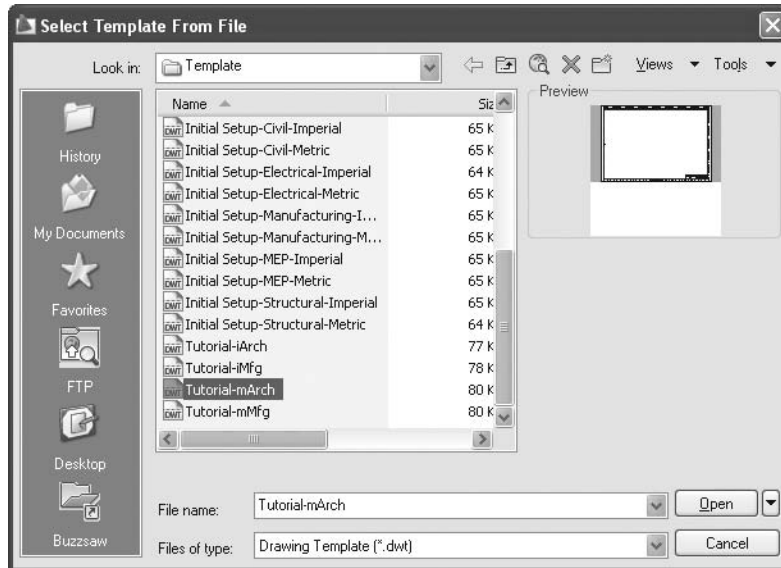
- There are several ways to create a new layout. You can select the **New Layout** option, use a template, or use the **Move or Copy** option.

New Layout

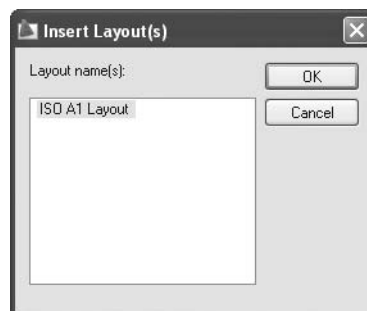
- New Layout** is a very simple method. First, right-click on any existing layout name and a shortcut menu will appear. Select the **New Layout** option.
- A new layout will be added with a temporary name. You can right-click and select the **Rename** option.

Using a Template

- You can import any layout defined inside a template and use it in your current drawing file.
- Right-click on any existing layout and a shortcut menu will appear. Select the **From template** option and the following dialog box will appear:



- Select the desired template and click **Open**. The following dialog box will appear:

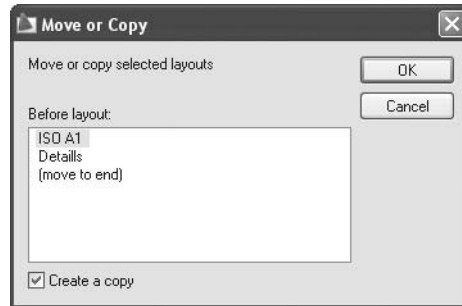


- Click on one of the listed layouts and click **OK**.

Move or Copy

- Using the **Move or Copy** option, you can move a layout to the left or right from its current position. Also, you can create a copy of an existing layout.

- Select the layout that you want to create a copy from and right-click. When the shortcut menu appears, select the **Move or Copy** option. The following dialog box will appear:

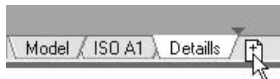


- You can see there are two existing layouts in the dialog box. Select one of them and click the **Create a copy** check box.
- Rename the new layout.
- Without using this command, you can move the layout position relative to the other layouts by clicking the layout name, holding it, and dragging it to the position required.



Copying Using the Mouse

- You can also copy any layout by performing the following steps:
 - Click the name of the layout to be copied.
 - Hold the [Ctrl] key on the keyboard.
 - Hold and drag the mouse to the new position of the newly copied layout.

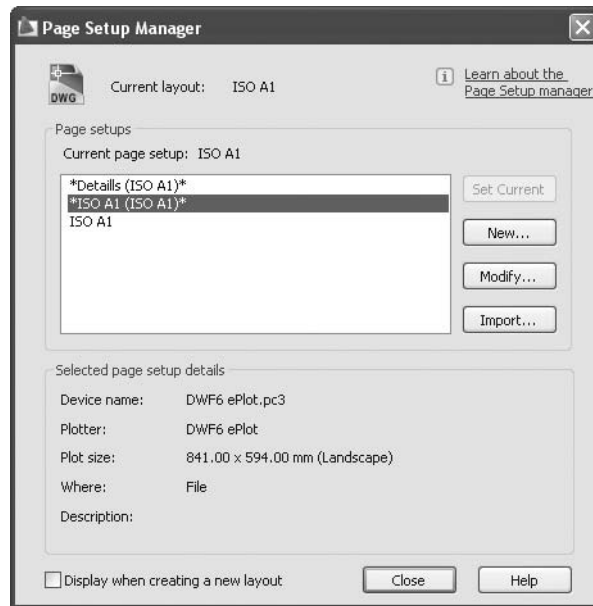


- Rename the new layout.

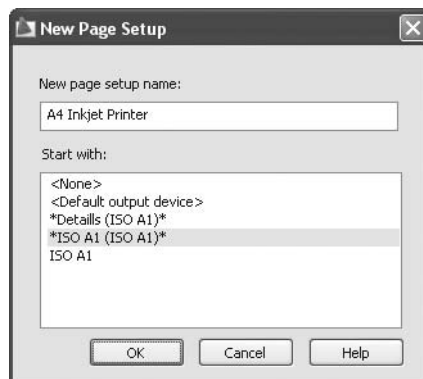
10.6 WHAT IS THE PAGE SETUP MANAGER?

- As mentioned previously, each layout will have a **Page Setup** linked to it.
- The **Page Setup Manager** is the dialog box in which you will create, modify, delete, and import **Page setups** for layouts.

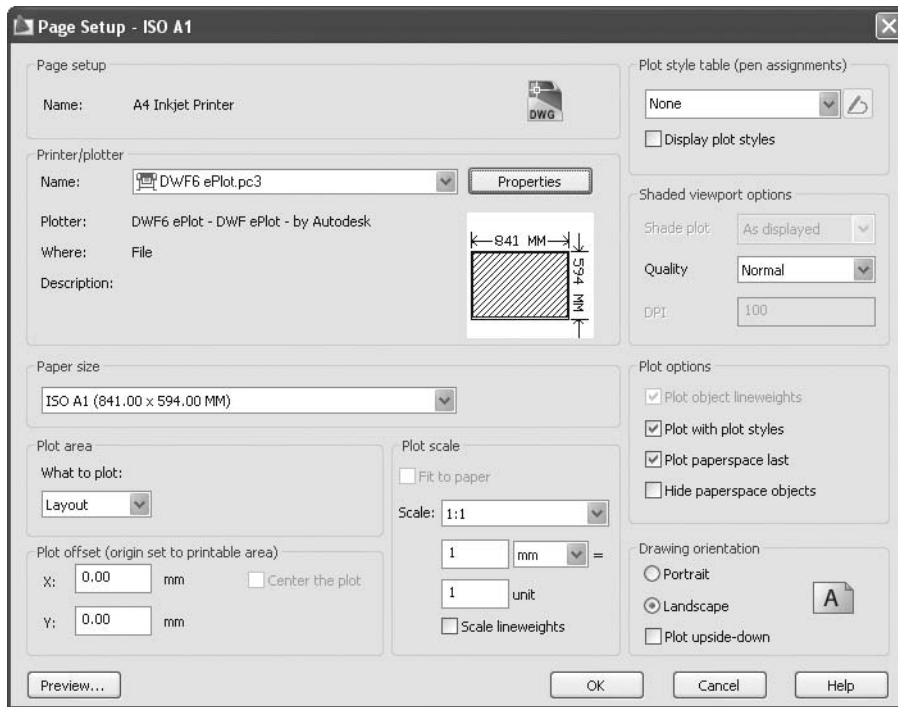
- The easiest way to issue this command is to select the desired layout, right-click, and then select **Page Setup Manager**.
- The following dialog box will appear:



- At the top, you will see the **Current layout** name, and at the bottom, you will see the **Selected page setup details**.
- A check box will allow you to **Display (Page Setup Manager) when creating a new layout**, which is highly recommended.
- To create a new **Page Setup**, click the **New** button. The following dialog box will appear:



- Type in the name of the new **Page Setup** and click **OK**. The following dialog box will appear:



- Specify the **Name** of the printer or plotter you want to use (this printer should be installed and configured ahead of time).
- Specify the desired **Paper Size**.
- Specify **What to plot**. You have four choices: **Display**, **Extents**, **Layout**, and **Window**. If you are printing from **Model Space** (this is not recommended), then choose **Display**, **Extents**, or **Window**. If you will print from **Layout** (which is recommended), leave **Layout** as the default setting.
- Specify the **Plot Offset**. If you are using **Layout**, **Center the plot** will be off. **Center the plot** will be on if you are using other options, such as **Model Space**.
- Specify the **Plot Scale**. If you want to plot from **Layout**, then you will use the **viewports** (which will be discussed shortly), and you will specify a **Plot scale** for each viewport. Accordingly, you will set this **Plot scale** to 1 = 1. Specify if you would like to **scale lineweights** or not.

- Specify the **Plot style table (pen assignments)**, which will be discussed later in this chapter. Specify whether to **Display** the effects of the plot style on the layout or not.
- If you are plotting a 3D drawing and you want to plot it as shaded or rendered, specify the **Quality** of the shading or rendering.
- Specify the **Plot options**. They are:
 - Plot the objects with their lineweight as specified for each object and layer. This will only be available if you specify **None** for the **Plot style** setting.
 - Let the **Plot Style** control the lineweight of the objects and layers.
 - By default, **Paper Space** objects will be printed first, and then the **Model Space** objects. You have the option to change the printing order.
 - You can show or hide the **Paper Space** objects.
- Specify the **Paper orientation**, either **Portrait** or **Landscape**. By default, the printer will start printing from top to bottom. Change the specifications if you would like to print from bottom to top.
- When you are done, click **OK**. The **Page Setup** you create will be available for all layouts in the current drawing file.
- To link any layout in your drawing file to a certain **Page Setup**, go to the desired layout and start the **Page Setup Manager**. Select the **Page Setup** from the list and click **Set Current**. (Also, you can double-click the name of the **Page Setup**.) Now the current layout is linked to the **Page Setup** you select.
- To modify the settings of an existing **Page Setup**, click **Modify**.
- To use a saved **Page Setup** from an existing file, click **Import**.

CREATING LAYOUTS AND PAGE SETUP (METRIC)



Workshop 19-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_19.dwg*.
3. Make sure the current layer is **Viewports**.
4. Using the **Status Bar**, right-click the **Layout1** icon, and select **Display Layout and Model tabs**.
5. Right-click on the name of any existing layout and select **From template**.
6. Select the template file *Tutorial-mArch.dwt*.

7. Select the layout named **ISO A1 Layout**.
8. Go to **ISO A1 Layout** and delete the only viewport in the layout (select its frame and press the [Del] key).
9. Delete **Layout2**.
10. Go to **Layout1**, rename it **Final**, and right-click the name of the layout. Select the **Page Setup Manager**.
11. Create a new **Page Setup** and name it **Final**.
12. Change the following settings:
 - a. **Printer** = DWF6 ePlot.pc3
 - b. **Paper Size** = ISO A3 (420 × 297 MM)
 - c. **Orientation** = Landscape
 - d. **Plot scale** = 1:1
13. Make **Final** the current page setup.
14. Erase the existing viewport.
15. Make the **Frame** layer current and insert the file with the name *ISO A3 Landscape Title Block.dwg* using 0,0 as the insertion point.
16. Save the file and close it.

CREATING LAYOUTS AND PAGE SETUP (IMPERIAL)



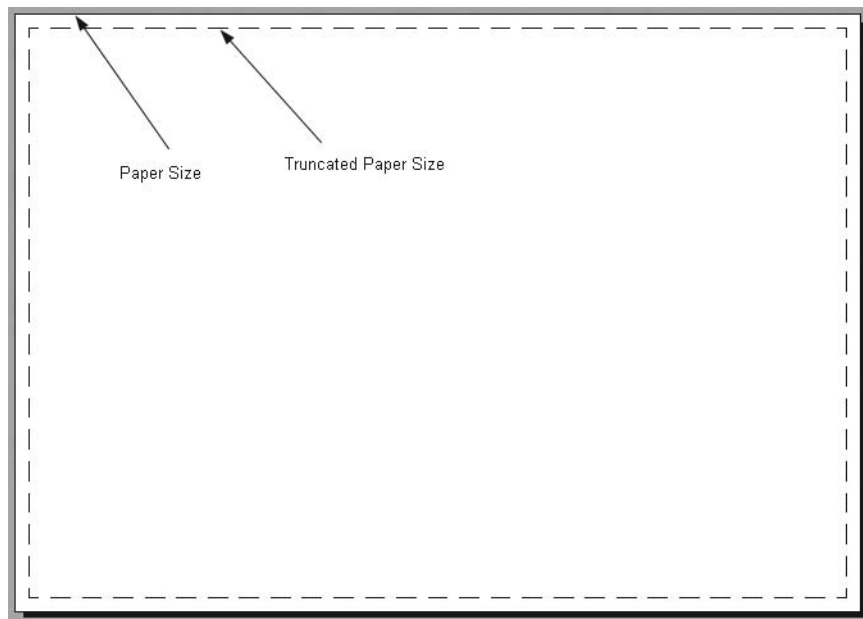
Workshop 19-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_19.dwg*.
3. Make sure the current layer is **Viewports**.
4. Using the **Status Bar**, right-click the **Layout1** icon, and select **Display Layout and Model tabs**.
5. Right-click on the name of any existing layout and select **From template**.
6. Select the template file *Tutorial-iArch.dwt*.
7. Select the layout name **D-Size Layout**.
8. Go to **D-Size Layout** and delete the only viewport in the layout (select its frame and press the [Del] key).
9. Delete **Layout2**.
10. Go to **Layout1**, rename it **Final**, and right-click the name of the layout. Select the **Page Setup Manager**.

11. Create a new **Page Setup** and call it **Final**.
12. Change the following settings:
 - a. **Printer** = DWF6 ePlot.pc3
 - b. **Paper Size** = ANSI B (17 × 11 Inches)
 - c. **Orientation** = Landscape
 - d. **Plot scale** = 1:1
13. Make **Final** the current page setup.
14. Erase the existing viewport.
15. Make the **Frame** layer current and insert the file with the name *ANSI B Landscape Title Block.dwg* using 0,0 as the insertion point.
16. Save the file and close it.

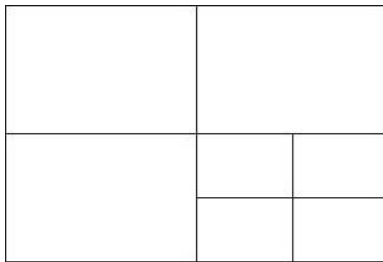
10.7 LAYOUTS AND VIEWPORTS

- After creating a new layout, creating a **Page Setup**, and linking a **Page Setup** to the layout, you will see the following image:

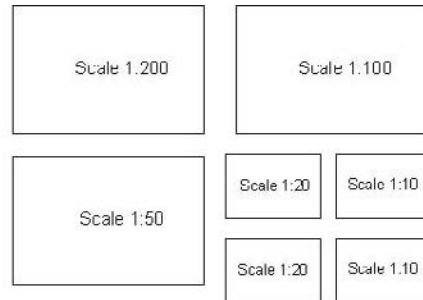


- The outer frame (solid) is the real paper size.

- The inner frame (dashed) is the truncated paper size, which is the paper size minus the printer margins.
 - Each printer comes from its manufacturer with built-in margins on all sides.
 - AutoCAD can read these margins from the printer driver accordingly.
 - Thus, you should read the printer manual in order to know exactly how wide the margins are on each side.
 - This will prove vital when you create the frame block of the establishment you will work in because you should create it within the truncated paper size rather than the full size.
- Printing from layouts is WYSIWYG (What You See Is What You Get).
- Also, by default, you will see that a single viewport of your drawing appears at the center of the paper size.
- As we said in the beginning of this chapter, we only have one **Model Space**, yet we can have as many **Layouts** as we wish. The **Viewport** is a rectangular shape (or any irregular shape) that contains a view of your **Model Space**.
- There are two types of **Viewports**: **Model Space** and **Layout**.
 - **Model Space Viewports**: These are always tiled, cannot be scaled, and the arrangement of viewports shown on the screen cannot be printed.
 - **Layout Viewports**: These can be tiled or separated, can be scaled, and the arrangement of viewports shown on the screen can be printed.
- See the following illustration:



Model Space Viewports



Paper Space Viewports

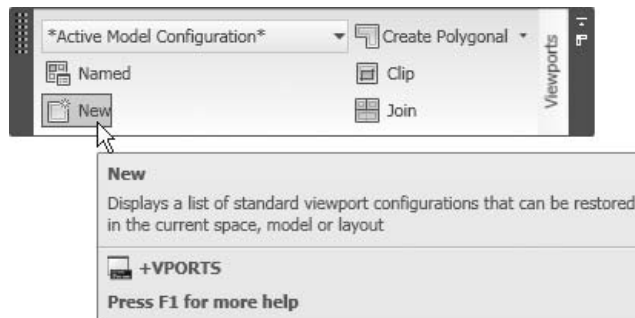
10.8 ADDING VIEWPORTS TO LAYOUTS

- You can add viewports to layouts using several different methods:
 - Adding a single rectangular viewport or multiple rectangular viewports.

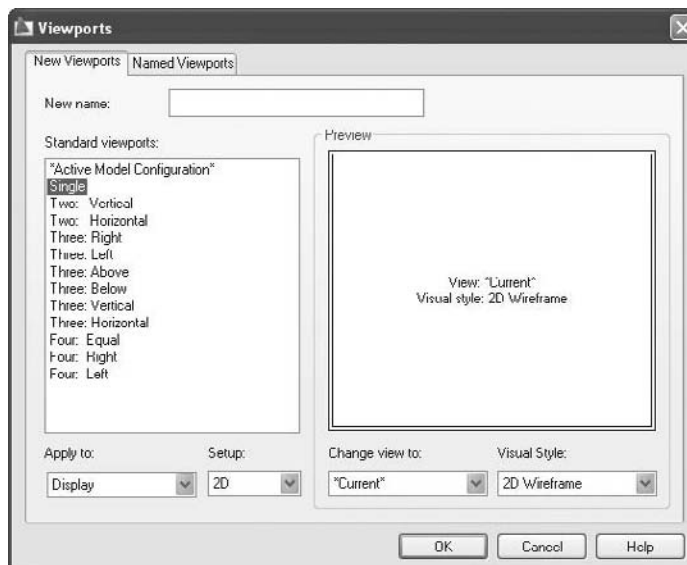
- Adding a single polygonal viewport.
 - Converting an object to be a viewport.
 - Clipping an existing viewport.
- We will discuss each method next.

Single or Multiple Rectangular Viewports

- You can add as many single or multiple rectangular viewports as you wish in any layout. You must specify two opposite corners in order to specify the area of the viewport(s).
- To start the **New Viewport** command, make sure you are in the **View** tab on the **Ribbon**, and, using the **Viewports** panel, click the **New** button.



- The following dialog box will appear:

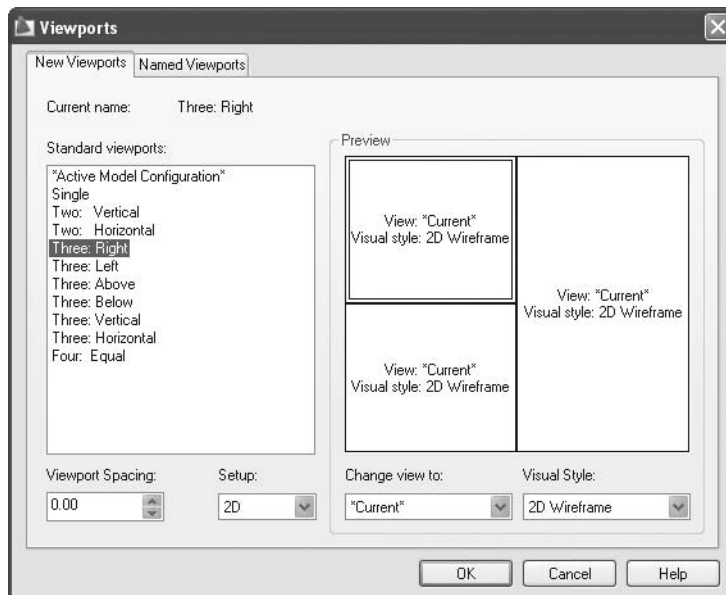


- From **Standard viewports**, select **Single** and then click **OK**. The following prompt will appear:

Specify first corner or [Fit] <Fit>:

Specify opposite corner:

- Just as we specify a window when we select objects, specify two opposite corners and a single viewport will be created.
- You will use the same dialog box:

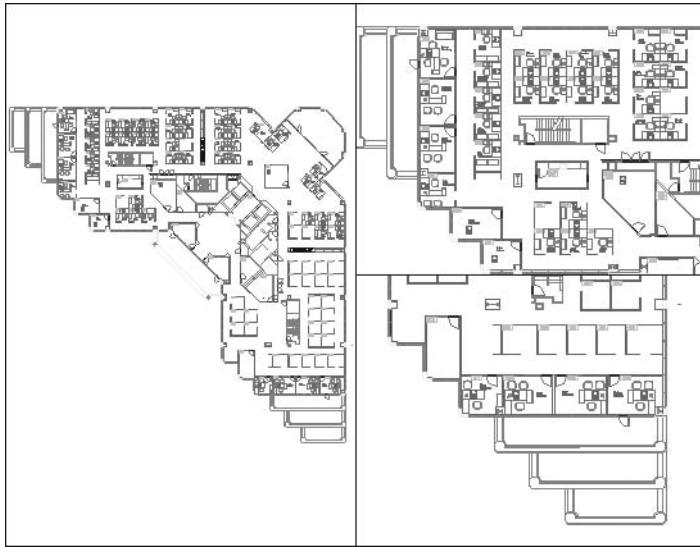


- Specify the desired arrangement. You can choose from **Two Horizontal** or **Two Vertical**, six different arrangements for three viewports, or one for four viewports.
- If you want the viewports to be tiled, leave the **Viewport Spacing** = 0. Otherwise, set a new value.
- Click **OK**. AutoCAD will display the following prompt:

Specify first corner or [Fit] <Fit>:

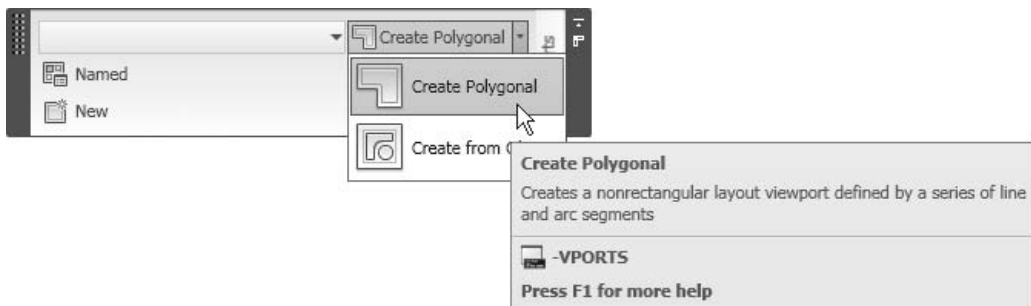
Specify opposite corner:

- See the following illustration:



Single Polygonal Viewport

- The **Polygonal Viewport** command is used to add a single viewport with any irregular shape consisting of both straight lines and arcs.
- To start the **Polygonal Viewport** command, make sure you are in the **View** tab on the **Ribbon**, and, using the **Viewports** panel, click the **Polygonal** button.



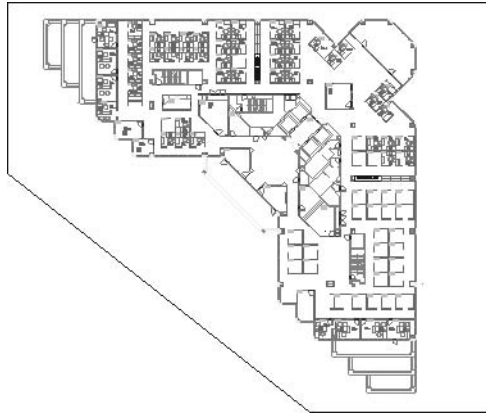
- The following prompt will appear:

```
Specify corner of viewport or
[ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/
Restore/Layer/2/3/4] <Fit>: _p
Specify start point:
```

Specify next point or [Arc/Length/Undo]:

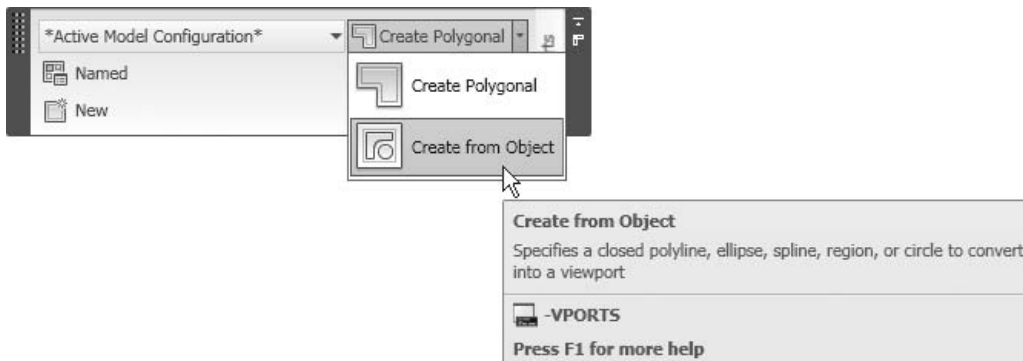
Specify next point or [Arc/Close/Length/Undo]:

- It is almost identical to the **Pline** command.
- See the following illustration:



Converting an Object to a Viewport

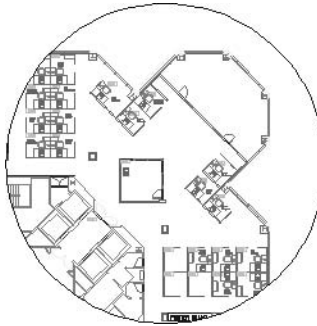
- The **Converting Object** command is used to convert an existing object to a viewport.
- First, you need to draw the object that will be converted to a viewport, such as a circle, polyline, ellipse, etc.
- To start the **Converting Object** command, make sure you are in the **View** tab on the **Ribbon**, and, using the **Viewports** panel, click the **Create from Object** button.



- The following prompt will appear:

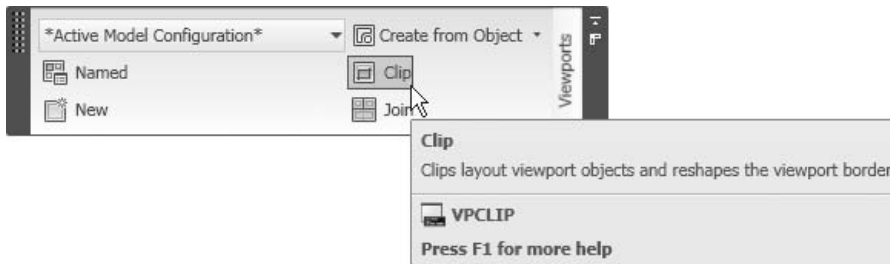
Specify corner of viewport or
 [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/
 Restore/Layer/2/3/4] <Fit>: _o
 Select object to clip viewport:

- See the following illustration:



Clipping an Existing Viewport

- If you have a rectangular viewport, you can change it to an irregular shape by clipping it.
- To start the **Viewport Clip** command, make sure you are in the **View** tab on the **Ribbon**, and, using the **Viewports** panel, click the **Clip** button.

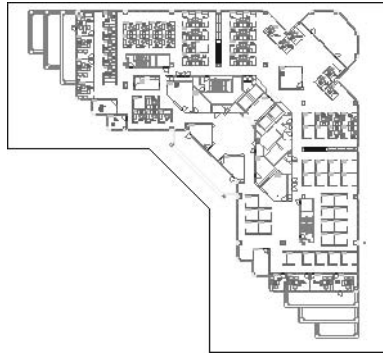


- The following prompt will appear:

Select viewport to clip:
 Select clipping object or [Polygonal] <Polygonal>:
 Specify start point:
 Specify next point or [Arc/Length/Undo]:
 Specify next point or [Arc/Close/Length/Undo]:

- First, select the existing viewport. You can draw a polyline or any irregular shape using the **Polygonal** option, which is identical to the **Polygonal viewport**.

- See the following illustration:



10.9 MODEL SPACE AND PAPER SPACE MODES IN LAYOUTS

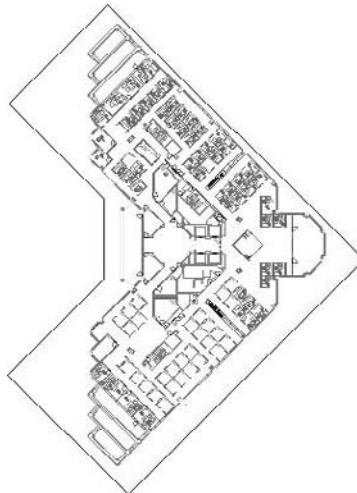
- In a layout, you deal with the viewports in two modes:
 - **Paper Space** mode
 - **Model Space** mode

Paper Space Mode

- The **Paper Space** mode is the default mode in any layout.
- In the **Status Bar**, you will see the following:



- As previously discussed, you can place the viewports in this mode.
- Also, you can deal with the viewports as objects; therefore, you can copy, move, stretch, rotate, and delete them. For example:



Rotated Viewport

Model Space Mode

- In **Model Space** mode you will get inside the viewport.
- You can zoom in, zoom out, and pan while you are in this mode.
- Also, you can scale each viewport.
- Furthermore, you can change the status of the layers for the current viewports.
- There are two ways to enter this mode:
 - Double-click inside the desired viewport.
 - Click the **Paper** button on the **Status Bar** and it will switch to **Model** as shown:



- In order to switch from **Model Space** mode to **Paper Space** mode, either double-click outside any viewport or click the **Model** button on the **Status Bar**.

10.10 MODIFYING, SCALING, AND MAXIMIZING VIEWPORTS

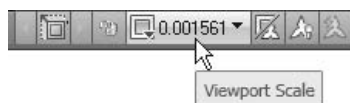
- Each viewport can be modified, scaled, or maximized to fill the whole screen.

Modifying

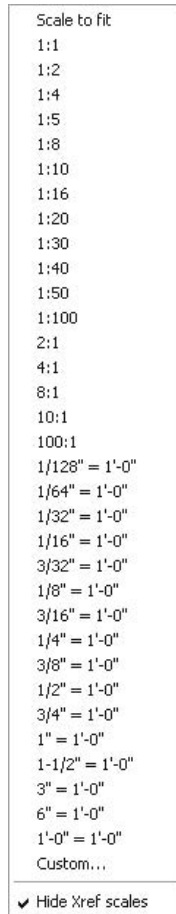
- Each viewport is considered an object. It can be copied, moved, scaled, rotated, and deleted. You have to select each viewport from its border in order to select it.
- You can select viewports first, and then issue the modifying command. Conversely, you can issue the command and then select the desired viewports.

Scaling

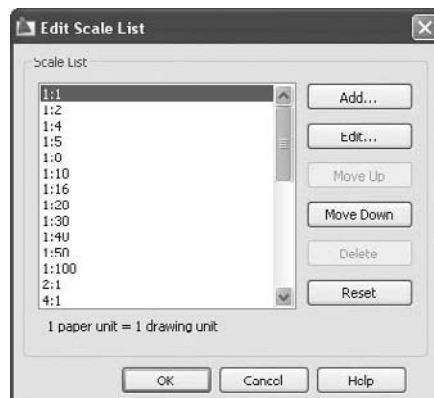
- Each viewport should be scaled relative to the **Model Space** units.
 - Double-click inside the desired viewport. You will switch to the **Model Space** mode for this viewport.
 - Look at the right side of the **Status Bar**, you will see the following:



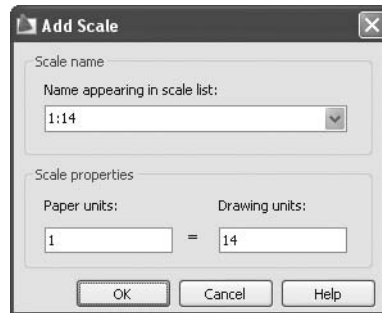
- Click the pop-up list that contains the scales. You will see something like the following:



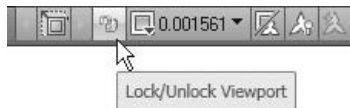
- Select the suitable scale to be used in the viewport.
- If you did not find the desired scale, select the **Custom** option. You will see the following dialog box:



- Click the **Add** button to add a new scale. You will see the following dialog box:



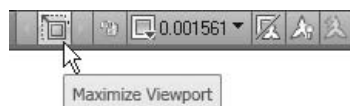
- Type in the desired scale and click **OK** twice.
- After you set the scale, you can use the **Pan** command. However, if you want to use the **Zoom** command, the scale value will be invalid and you will have to repeat the procedure of setting the scale again.
- In order to avoid this problem, you can lock the display of the viewport by clicking the golden opened lock in the **Status Bar** (you have to be inside the viewport in order for this to work). The golden lock will change to blue and it will be locked.



- There are two possible results once you have scaled a viewpoint:
 - The scale is perfect for the area of the viewport. Leave it as is.
 - The scale is either too small or too big. You can change the scale or change the area of the viewport.

Maximizing

- After placing and scaling your viewports, there will be small ones and big ones.
- For small ones, you can maximize the area of the viewport to be as large as your screen. You can do all of your work and then return it to the original size.
- Using the **Status Bar**, click the **Maximize Viewport** button:

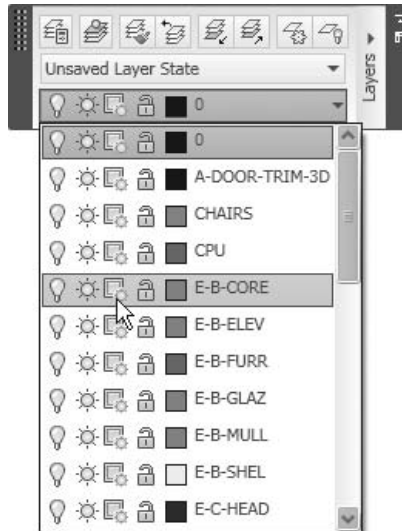




- The same button will change to **Minimize Viewport** in order to restore the original size of the viewport.
- Another way of maximizing the viewport is to double-click the border of the viewport.

10.11 FREEZING LAYERS IN VIEWPORTS

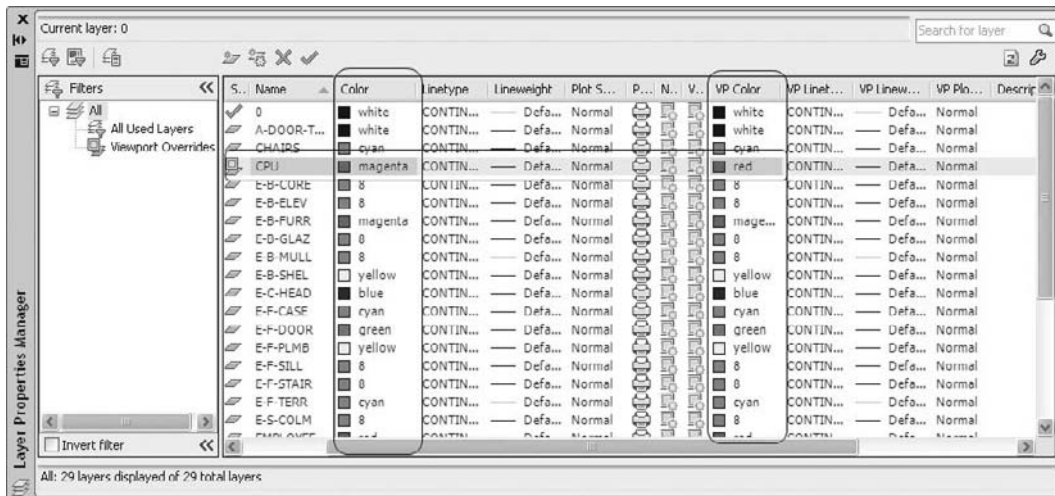
- Earlier in this book we learned how to freeze a layer. This tool will be effective in both **Model Space** and **Layouts**.
- In **Layouts**, if you freeze a layer it will be frozen in all viewports. You may want to freeze certain layer(s) in one of the viewports and not in others. To do so, you have to freeze the layer in the current viewport.
- Perform the following tasks:
 - Make the desired viewport current by double-clicking inside it.
 - Make sure you are in the **Home** tab on the **Ribbon**, and, using the **Layers** panel, click the **Freeze or thaw in current viewport** icon for the desired layer. See the following image:



10.12 LAYER OVERRIDE IN VIEWPORTS

- **Color**, **Linetype**, **Lineweight**, and **Plot Style** are the same for all layers in all viewports.
- You can assign different colors, linetypes, lineweights, or plot styles for each layer in each viewport. This is called **Layer Override**.

- Perform the following steps:
 - Double-click inside the desired viewport.
 - Start the **Layer Properties Manager**.
 - Under **all/any** of **VP Color**, **VP Linetype**, **VP Lineweight**, or **VP Plot Style**, make the desired changes.
 - You will see that changes only take place in the current viewport.
 - See the following image:



- You can see that the **CPU** layer has a magenta normal color and a red override color in the current viewport (this applies in **Model Space** and all other viewports).
- Also, note that the row containing the **CPU** layer is shaded with a different color.

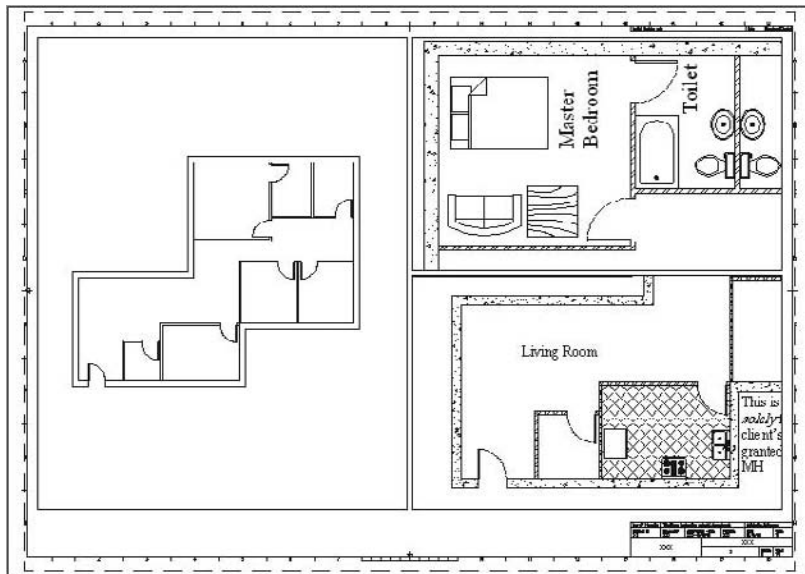
INSERTING AND SCALING VIEWPORTS (METRIC)



Workshop 20-A

1. Start AutoCAD 2010.
2. Open the file *Workshop_20.dwg*.
3. Select **ISO A1 Layout**.
4. Make the **Viewports** layer current.
5. Turn **OSNAP** off.
6. Select the **View** tab from the **Ribbon**, and, using the **Viewports** panel, click the **New** button. Select the arrangement **Three: Left**, and set the **Viewport spacing** = 5.

7. Click **OK** and specify two opposite corners so the three viewports fill the empty space.
8. Select the big viewport at the left and set the scale = 1:50, the upper viewport scale = 1:20, and the lower viewport scale to be 1:30.
9. Double-click outside the viewports to move to the **Paper** mode.
10. Freeze the **Dimension** layer and thaw **Furniture**, **Hatch**, and **Text**.
11. Make the big left viewport the current viewport, and freeze the layers **Furniture**, **Hatch**, and **Text**.
12. Make the upper viewport current and pan to the Master Bedroom (do not use the zooming options). Make the lower viewport current and pan to the Living Room.
13. Double-click outside the viewports to move to the **Paper** mode.
14. Lock the view in the three viewports.
15. The drawing should look like the following:



16. Save the file and close it.

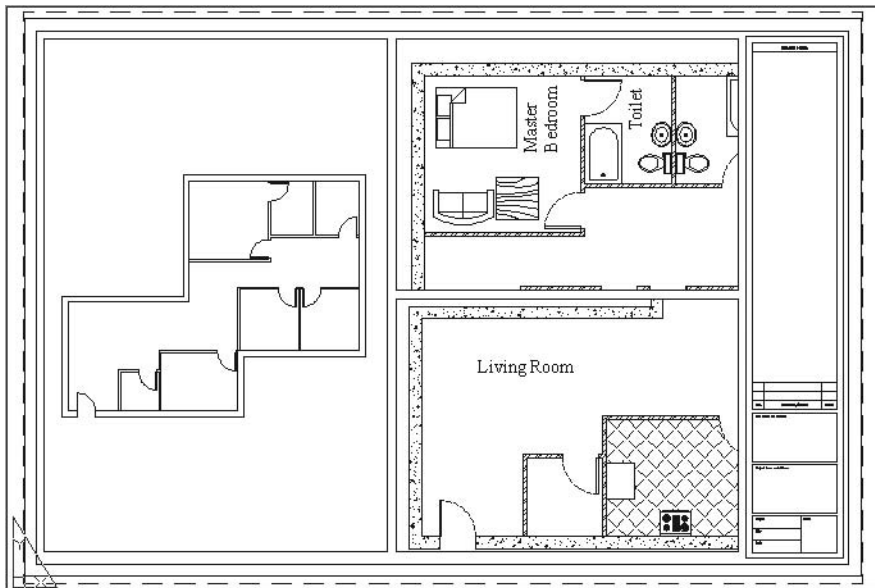
INSERTING AND SCALING VIEWPORTS (IMPERIAL)



Workshop 20-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_20.dwg*.

3. Select **D-Sized Layout**.
4. Make the **Viewports** layer current.
5. Turn **OSNAP** off.
6. Make sure you are in the **View** tab on the **Ribbon**, and, using the **Viewports** panel, click the **New** button. Select the arrangement **Three: Left**, and set the **Viewport spacing** = 0.35.
7. Click **OK** and specify two opposite corners so the three viewports fill the empty space.
8. Select the big viewport at the left and set the scale to be $\frac{1}{4}" = 1'$, and the upper and lower viewports scale to be $\frac{1}{2}" = 1'$.
9. Double-click outside the viewports to move to the **Paper** mode.
10. Freeze the layer **Dimension** and thaw **Furniture**, **Hatch**, and **Text**.
11. Make the big left viewport the current viewport and freeze the layers **Furniture**, **Hatch**, and **Text**.
12. Make the upper viewport current and pan to the Master Bedroom (do not use the zooming options). Make the lower viewport current and pan to the Living Room.
13. Double-click outside the viewports to move to the **Paper** mode.
14. Lock the view in the three viewports.
15. The drawing should look like the following:



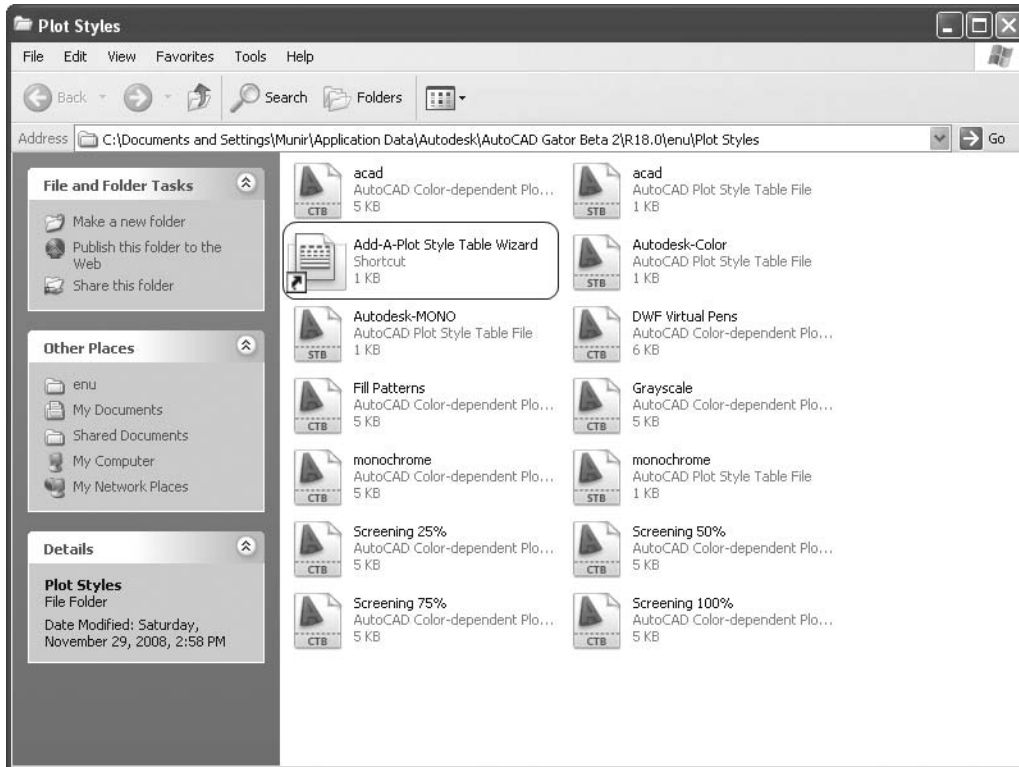
16. Save the file and close it.

10.13 AN INTRODUCTION TO PLOT STYLE TABLES

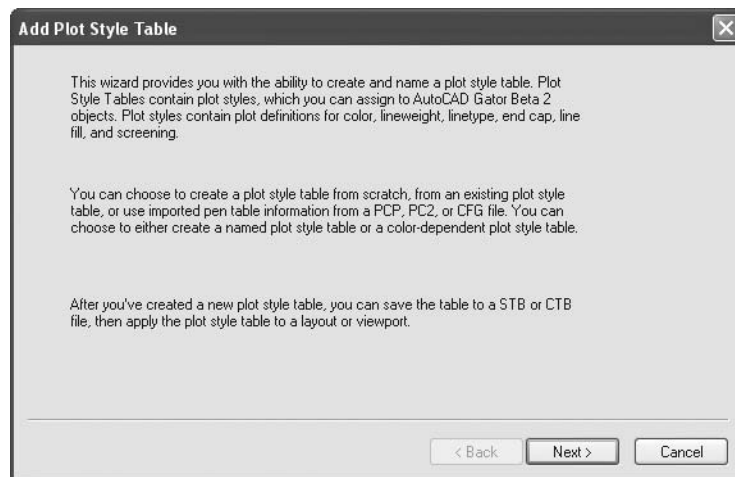
- There are many colors available in AutoCAD, but, will these colors print?
- There are two possibilities:
 - You will use the same colors in both the softcopy and the hardcopy of the drawing.
 - You will assign a different color in the hardcopy for each color in the softcopy.
- To translate the colors between softcopy and hardcopy, we need to create a **Plot Style**.
- There are two types of **Plot Styles**:
 - **Color-Dependent Plot Style Table**
 - **Named Plot Style Table**

10.14 THE COLOR-DEPENDENT PLOT STYLE TABLE

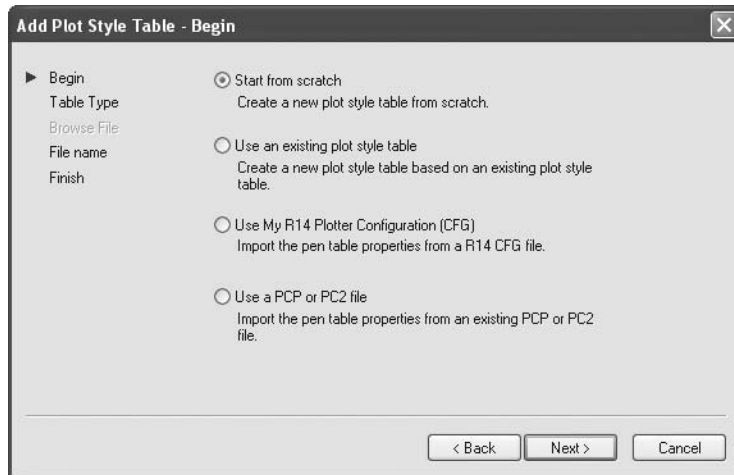
- This method is almost the same method used prior to AutoCAD 2000; it depends on the colors used in the drawing file.
- Each color used in the drawing file will be printed with a color chosen by the user. Also, users can set the lineweight, linetype, etc., for each color.
- This method is limited because there are only 255 colors to use.
- Also, if there are two layers with the same color, you will be forced to use the same output color, with the same lineweight, linetype, etc.
- Each time you create a **Color-Dependent Plot Style Table**, AutoCAD will ask you to name a file with the extension **.ctb*.
- You can create **Plot Style Tables** from outside AutoCAD (using the **Windows Control Panel**) or from inside AutoCAD using the **Wizards**. This will only initiate the command, but the command is the same either way.
- From outside AutoCAD, start the **Windows Control Panel** and double-click the **Autodesk Plot Style Manager** icon. Since this command is not available on the **Ribbon**, you can type **stylesmanager** in the **Command Window**.
- You will see the following window:



- Double-click **Add-A-Plot Style Table Wizard**.
- The following dialog box will appear:



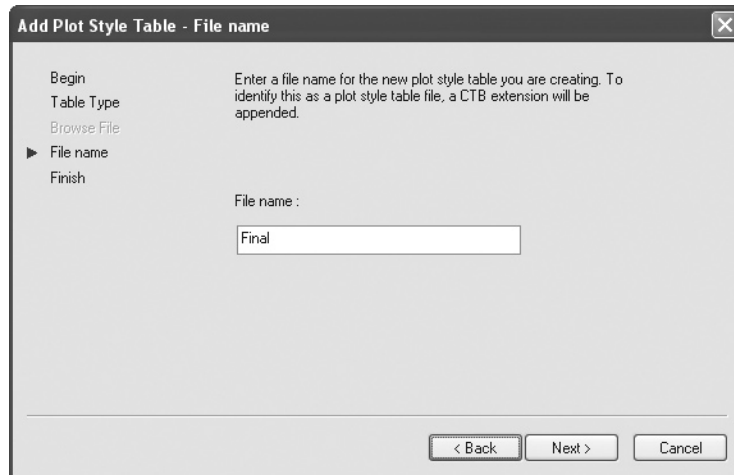
- In this dialog box, AutoCAD explains the next few steps in this process. Click **Next** and the following dialog box will appear:



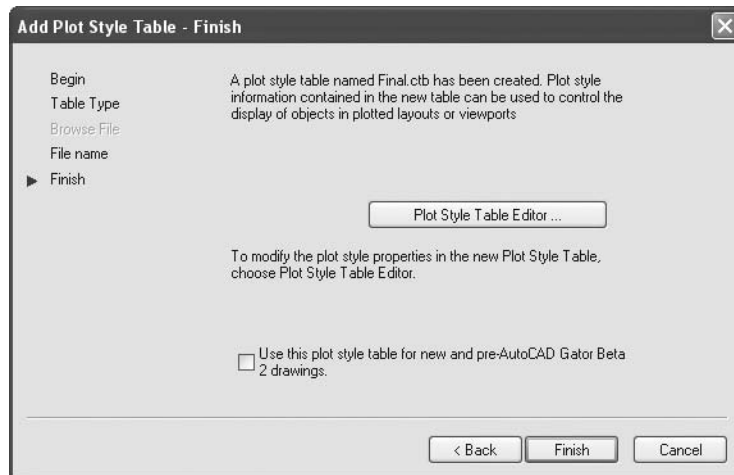
- You have four choices to select from:
 - Start creating your style from scratch.
 - Use an existing plot style.
 - Import the *AutoCAD R14 CFG* file and create a plot style from it.
 - Import the *PCP* or *PC2* file and create a plot style from it.
- Select **Start from scratch** and click **Next**. The following dialog box will appear:



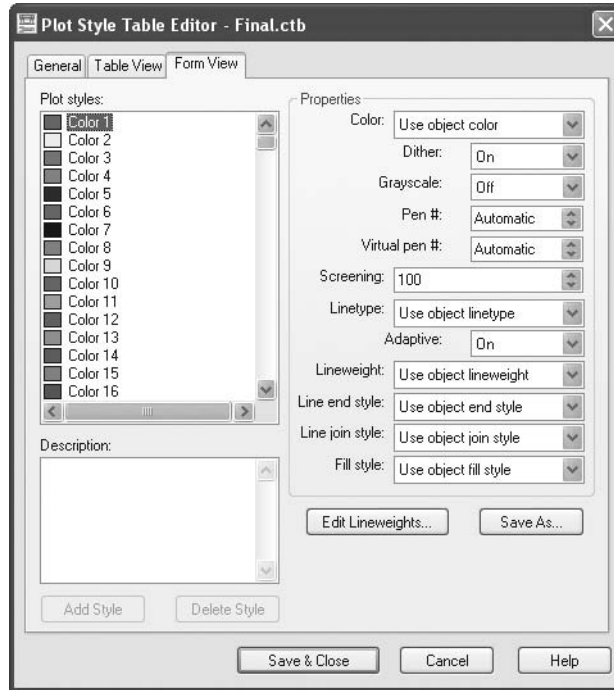
- Select **Color-Dependent Plot Style Table** and click **Next**.
- The following dialog box will appear:



- Type in the name of the plot style and click **Next**. The following will appear:

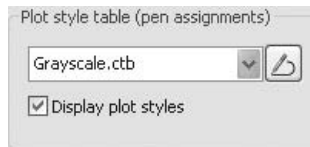


- You can use this plot style table for new and pre-AutoCAD 2010 drawings. Click the **Plot Style Table Editor** button and you will see the following:

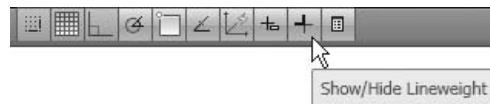


- At the left, select the color you used in your drawing file. At the right, change any or all of the following settings.
 - **Color:** This is the hardcopy color.
 - **Dither:** This option will be dimmed if your printer or plotter does not support dithering. Dither allows the printer to give the impression of using more colors than the limited 255 colors provided by AutoCAD. It is preferable to leave this option off. It should be on if you want **Screening** to work.
 - **Grayscale:** You can translate the 255 colors to grayscale grades (good for laser printers).
 - **Pen #:** This option is only for the old types of plotters—pen plotters—that are obsolete now.
 - **Virtual pen #:** This option is for non-pen plotters to simulate pen plotters by assigning a virtual pen to each color. It is preferable to leave this setting on **Automatic**.
 - **Screening:** This is good for trial printing. If you set numbers less than 100, it will reduce the intensity of the shading and fill hatches. You should turn **Dither** on so **Screening** will be effective.
 - **Linetype:** You can use the object's linetype or you can set a different linetype for each color.

- **Adaptive:** Used to change the linetype scale to fit the current line length so it will start with a segment and end with a segment instead of a space. Turn this option off if the linetype scale is important to the drawing.
 - **Lineweight:** Sets the lineweight for the color selected. You can choose from a list of lineweights.
 - **Line end style:** Used to specify the end style for lines. The available end styles are: **Butt**, **Square**, **Round**, and **Diamond**.
 - **Line join style:** Used to specify the line join (the connection between two lines) style. The available styles are: **Miter**, **Bevel**, **Round**, and **Diamond**.
 - **Fill style:** Used to set the fill style for the area filled in the drawing, which is good for trial printing.
- Click **Save & Close** and then click **Finish**.
 - To link **Color-Dependent Plot Style** to a layout, perform the following steps:
 - Go to the desired layout and start the **Page Setup Manager**.
 - At the upper right section of the dialog box, change the **Plot style table (pen assignments)** setting to the desired *.ctb* file:



- Click the **Display plot styles** checkbox.
- You can assign one *.ctb* file for each layout.
 - In order to see the lineweight of the objects, you must click on the **Show/Hide Lineweight** button on the **Status Bar**.



10.15 THE NAMED PLOT STYLE TABLE

- This is a new method introduced for the first time in AutoCAD 2000; it is not color-dependent.
- You will create a **Plot Style** and give it a name. Each plot style will include different tables within it, which you will link with layers later on.
- You can have two layers with the same color, but they will be printed with different colors, linetypes, and lineweights.

- The **Named Plot Style Table** has the file extension *.*stb*.
- The creation procedure of the **Named Plot Style** is identical to that of the **Color-Dependent Plot Style**, except the last step, which uses the **Plot Style Table Editor** button.
- From outside AutoCAD, start the **Windows Control Panel** and double-click the **Autodesk Plot Style Manager** icon. Since this command is not available on the **Ribbon**, you can type **stylesmanager** in the **Command Window**.
- Double-click **Add-A-Plot Style Table Wizard**.
- Go through the dialog boxes until you reach the **Plot Style Table Editor** button. Click it and you will see the dialog box that follows.
- After you click the **Add Style** button, you will see the **Plot Style Table Editor** for the **Named Plot Style**:

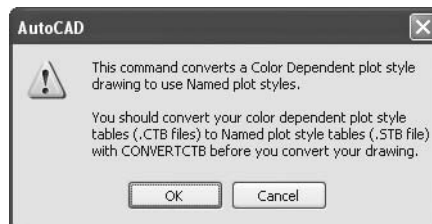


- As you can see, you can change the **Name**, **Description**, or **Color**. The rest is identical to the **Color-Dependent Plot Style Table**. To change any of the elements listed:
 - Type in the **Name** of the style.
 - Type in any **Description** for this style.
 - Specify the **Color** that you will use in the hardcopy.
- You can add as many styles as you wish in the same **Named Plot Style**.
- Click **Save & Close** and then click **Finish**.
- In order to link a **Named Plot Style Table** with a drawing:

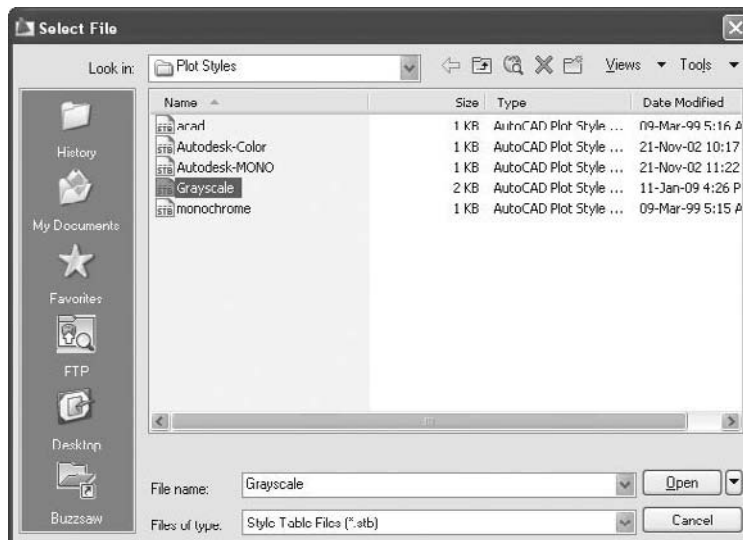
- You must first convert one of the *.ctb* files to a *.stb* file. Type **convertctb** in the **Command Window**. A dialog box with all the *.ctb* files will appear. Select one of them, keeping the same name or giving a new name, then click **OK**. The following dialog box will appear:



- Convert the drawing from **Color-Dependent Plot Style** to **Named Plot Style**. Type **convertplotstyles** in the **Command Window**. The following warning message will appear:



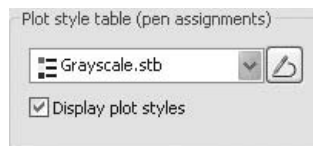
- Click **OK** and the following dialog box will appear:



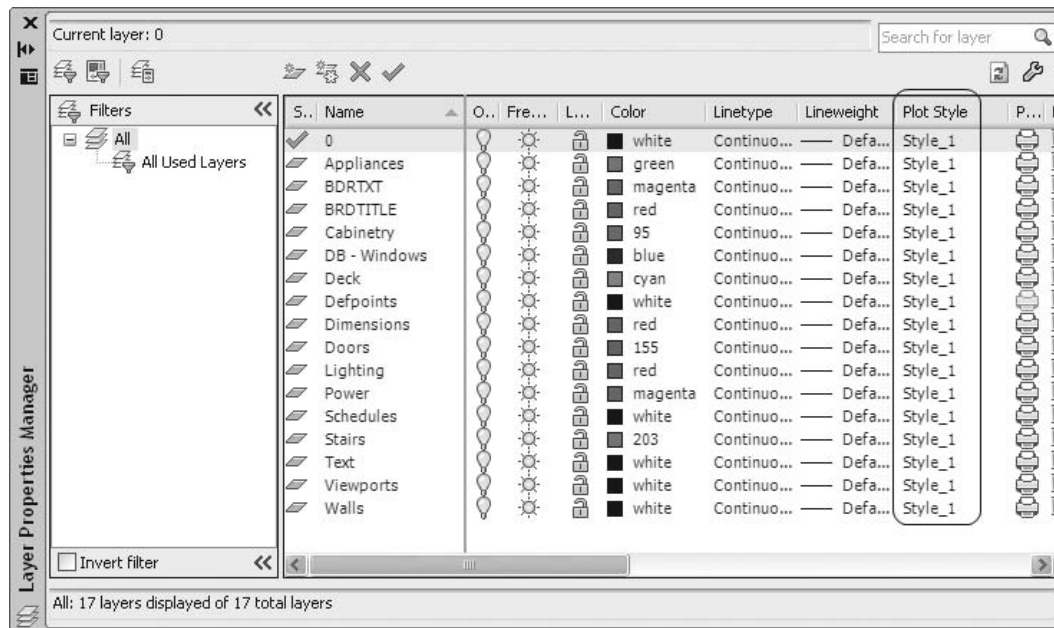
- Select the **Named Plot Style Table** you just converted and click **Open**. The following message will appear in the Command Window:

Drawing converted from Color Dependent mode to
Named plot style mode.

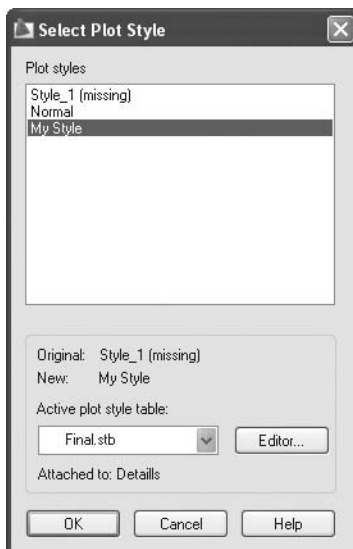
- Because you will use the **Named Plot Style Table** with layers, the **Model Space** and all layouts will be assigned the same *.stb* file.
- Go to the desired layout and start the **Page Setup Manager**. In the upper right-hand part of the dialog box, change the **Plot style table (pen assignments)** setting to the desired *.stb* file. Click the **Display plot styles** checkbox and end the **Page Setup Manager** command:



- Go to the **Layer Properties Manager** and search for the desired layer under the **Plot Style** column:



- Click the name of the current style. The following dialog box will appear:



- Select the desired plot style to be linked to the selected layer. When you are finished, click **OK**.

PLOT STYLE TABLES



Exercise 40

1. Start AutoCAD 2010.
2. Open the file *Exercise_40.dwg*.
3. Click **Layout1**.
4. Start a new **Plot Style Table** and choose **Color-Dependant**. Call this plot style **Mechanical_BW** and make the following changes:

Drawing Color	Plotter Color	Linetype	Lineweight
Color 2	Black	Dashdot	0.30
Color 3	Black	Solid	0.70
Color 4	Black	Solid	0.50
Color 6	Black	Solid	0.50

5. Go to the **Page Setup Manager** and select **Plot style table** to be **Mechanical_BW**. Turn on **Display plot styles**.

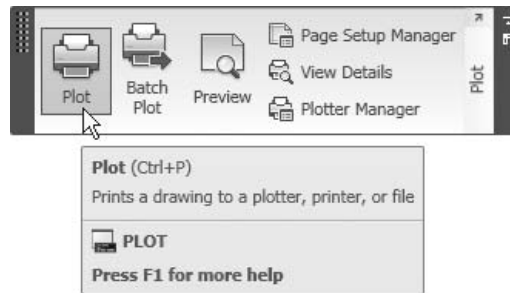
6. Click the **Show/Hide Lineweight** button to see the effect of the lineweight.
7. Save the file as *Exercise_40_1.dwg*.
8. Type **convertctb** in the **Command Window** and convert **Mechanical_BW** from a *.ctb* to a *.stb* file.
9. Type **convertstyles** in the **Command Window** to convert the whole drawing to accept **Named Plot Styles**. Select **Mechanical_BW** when you are asked to select a *.stb* file.
10. Start a new **Plot Style Table** and choose **Named Plot Style**. Call this plot style **Design_Process** and make the following changes:

Style	Description	Color	Linetype	Lineweight
Finished	Design is Final	Black	Solid	0.7
Incomplete	Incomplete Design	Green	Dashed	0.3

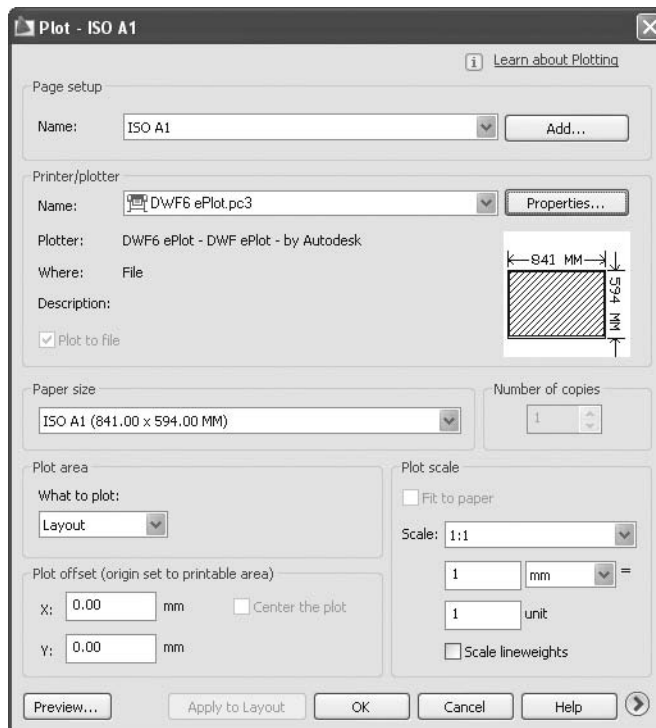
11. Go to the **Page Setup Manager** and select **Plot Style Table** to be **Design_Process**. Turn on **Display plot styles**.
12. Start the **Layer Manager** and set the plot style for layer **Base** to be **Finished**, and layers **Shaft** and **Body** to be **Incomplete**.
13. Click the **Show/Hide Lineweight** button to see the effect of the lineweight.
14. Issue the **Regenall** command if things do not appear correctly.
15. Save the file as *Exercise_40_2.dwg*.

10.16 THE PLOT COMMAND

- The final step in this process is to issue the **Plot** command, which will send your layout to the printer or plotter.
- As a first step, go to the desired layout you want to plot.
- To issue this command, make sure you are in the **Output** tab on the **Ribbon**, and, using the **Plot** panel, click the icon **Plot**.

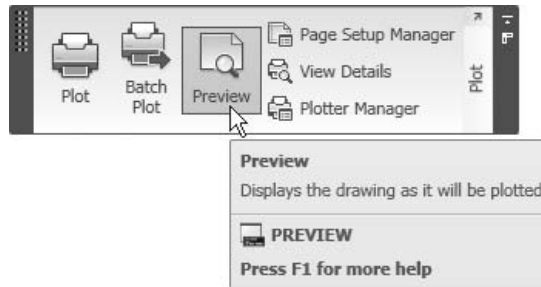


- The following dialog box will appear:



- As you can see, all of the settings are identical to those in **Page setup**.
- If you change any of these settings, AutoCAD will detach the **Page setup** from the current layout.
- Click the **Apply to Layout** button if you want this plot dialog box saved with this layout for future use.

- Click the **Preview** button in order to see the final printed drawing on the screen before the real printout. You can decide if your choices of plot styles and other settings are correct or not.
- You can preview your drawing from outside this dialog box. Make sure that you are in **Output** tab on the **Ribbon**, and, using the **Plot** panel, click the **Preview** button.



- After you are done, click **OK**. The drawing will be sent to the printer.

10.17 WHAT IS A DWF FILE?

- Assume one or all of the following:
 - You want to share your design with another company, but you are fearful that if you send them the *.dwg* file they will alter it.
 - Your *.dwg* file is very large (more than 1 MB), which may not be accepted by your email server.
 - The recipient does not have AutoCAD to view the *.dwg* file.
- AutoCAD offers you the option of plotting to a DWF file (Design Web Format). This file has the following features:
 - You do not need AutoCAD to open a DWF file. Free software comes with AutoCAD called **Autodesk Design Review**, which you can also download from the Internet free of charge.
 - You can view the file, zoom, pan, measure, markup, and print it.
 - The size is small so you can send it through email.
 - The recipient cannot modify it.

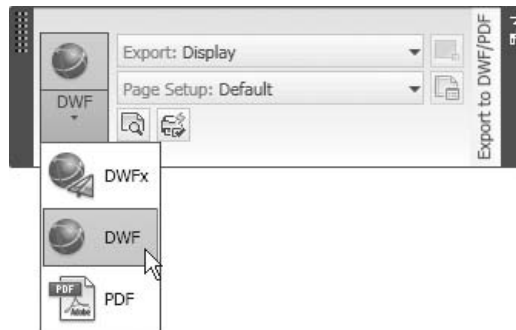
10.18 WHAT IS A DWFx FILE?

- A DWFx file was introduced by both Autodesk®, Inc., and Microsoft®.
- It will replace the DWF as the next generation file.

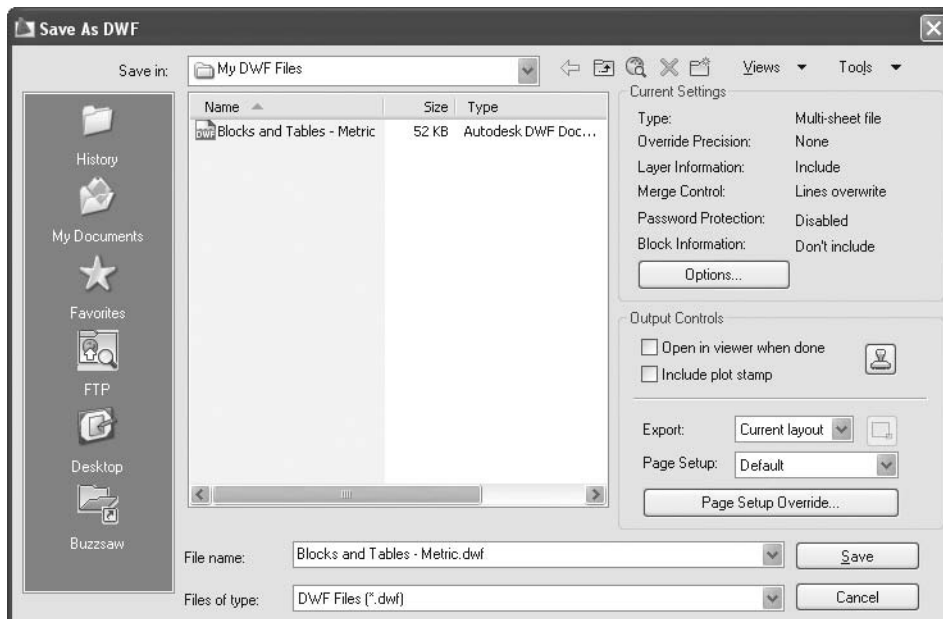
- If you are using Windows Vista, you will be able to view a DWFx file without any additional viewer.
- DWFx can be viewed using Windows Internet Explorer.

10.19 EXPORTING DWF, DWFx, AND PDF FILES

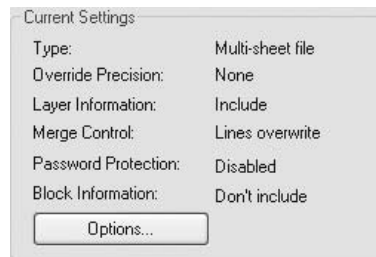
- You can export DWF, DWFx, and PDF files using AutoCAD 2010.
- To start exporting, make sure you are in the **Output** tab on the **Ribbon**, and, using the **Export to DWF/PDF** panel, click any of the following three buttons:



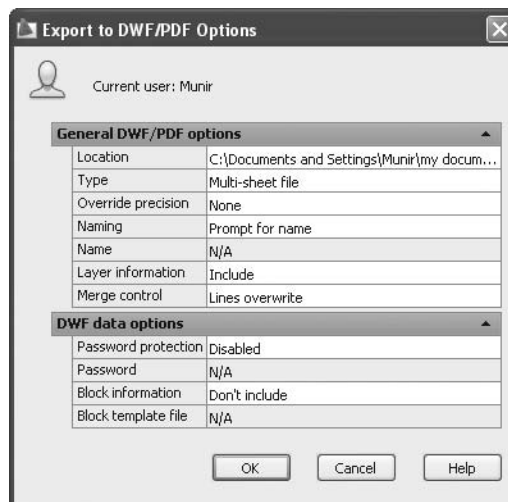
- The dialog box for the three types is identical:



- Specify the hard disk, folder, and the name of the file.
- Read the **Current Settings** at the upper right-hand part of the dialog box:

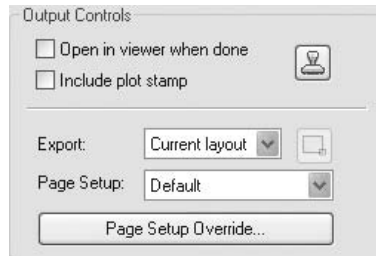


- If these are the settings you want, go to the next step. If not, click the **Options** button and you will see the following dialog box:



- Adjust the following settings:
 - Specify the location of the file.
 - Decide if the file will be **Single sheet** (current layout) or **Multi-sheet** (all layouts). While exporting, it will always produce a single layout that is the current layout. This option will be discussed along with the **Publish** command.
 - If you select a **Multi-sheet file**, determine whether or not to **Prompt for name**.
 - Select whether or not to include the **Layer information** in the file.
 - Select whether or not to include a **Password** for the DWF file. This option is not available for the PDF file.
 - Select whether or not to include the **Block information** (like attributes) in the file.

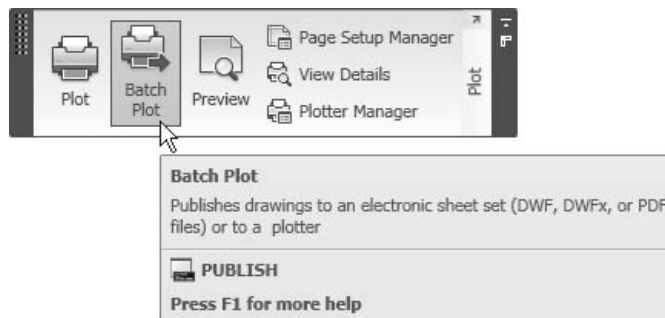
- Once you are done, click **OK**.
- Under the **Output Controls**:



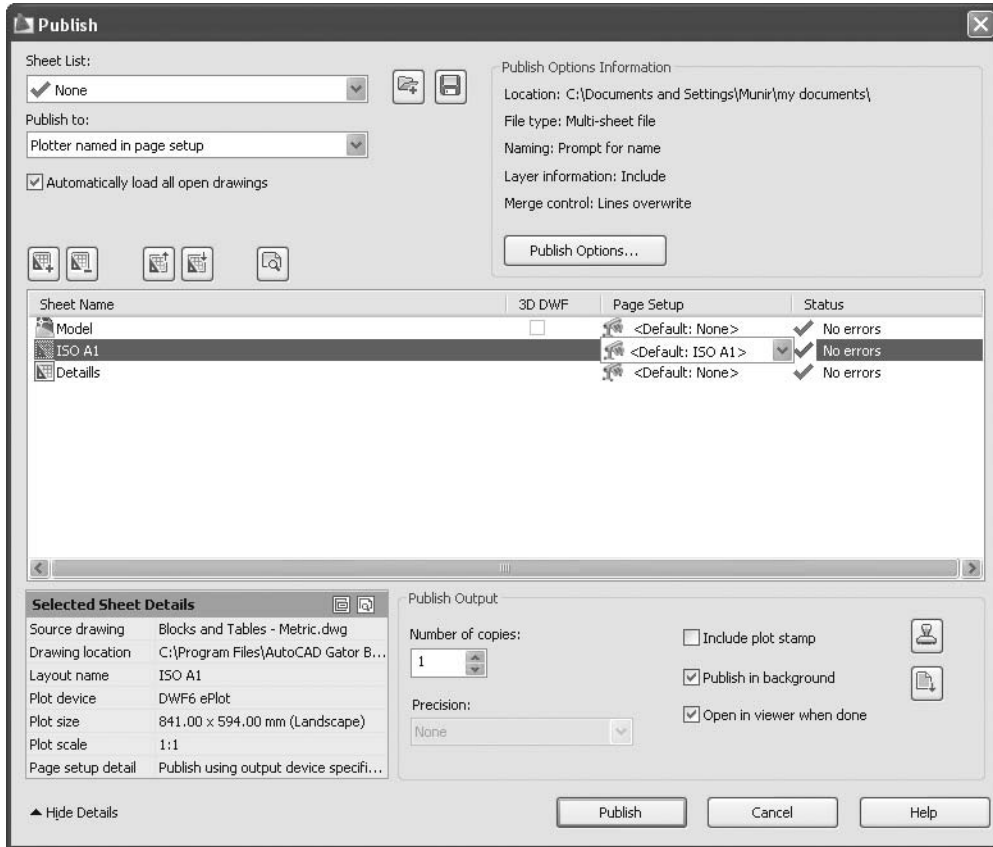
- Adjust the following:
 - Select whether to open the file in the viewer when done or not.
 - Select whether to include a **plot stamp** or not.
 - Select what to export. If you are at a layout, **Current layout** will be selected automatically.
 - Select the **Page Setup**. AutoCAD will select the page setup included inside the layout. You can select any other page setups that exist in the drawing or you can create a temporary page setup for this plot only.
- When you are done, click the **Save** button.

10.20 THE PUBLISH COMMAND

- The **Publish** command will produce a DWF file containing multiple layouts from the current drawing and from other drawings.
- To issue the **Publish** command, make sure you are in the **Output** tab on the **Ribbon**, and, using the **Plot** panel, click the **Batch Plot** button:



- The following dialog box will appear:



- You will see a list of the current file's **Model Space** and layouts.
- You can specify a previously saved **Sheet list** in the current drawing or you can:



- Click the open folder icon to open a previously saved **Sheet list**.



- Click the disk icon to save a **Sheet list** for future printing.

- Select whether you want to use the printer/plotter defined in the layout or a DWF, DWFx, or PDF.
- Select whether to load all open drawings automatically or not.

- Select one of the sheets and use the following buttons:
 - This button adds more sheets from other drawings. The **Select Drawings** dialog box will be shown to select the desired file.
 - This button removes one or more sheets from the list.





- This button moves the sheet up the list.

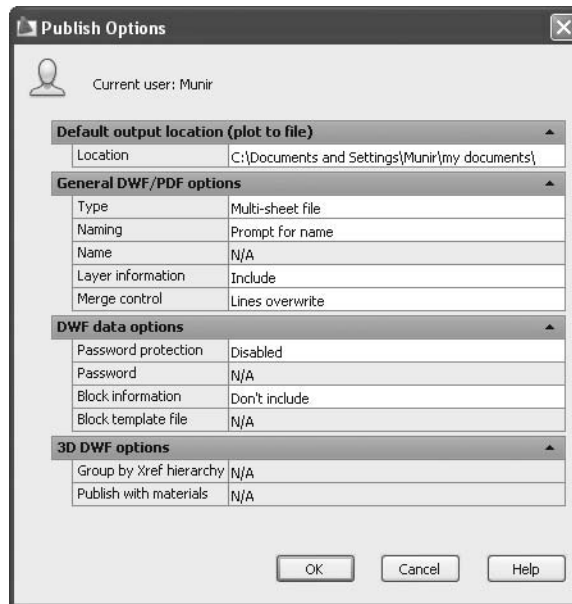


- This button moves the sheet down the list.

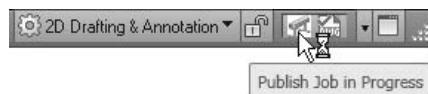


- Use this button to preview the selected sheet (only single sheet) as we did in **Print Preview**.

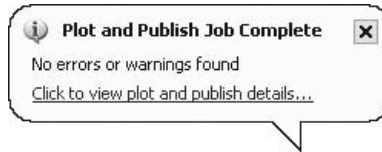
- Click **Publish Options** and you will see the following dialog box:



- This dialog box was discussed in the section on DWF, DWFx, and PDF exporting.
- **Multi-sheet file** means all sheets in the current sheet list.
- Specify the number of copies.
- To include a **Plot Stamp** in each sheet, check **Include plot stamp**.
- Select whether you want to publish in the background or not.
- Select whether to open the file in the viewer when done or not.
- You can see more details about the selected sheet, such as **Plot device**, **Plot size**, **Plot scale**, etc.
- Once you are done setting all of these, click the **Publish** button. AutoCAD will create the sheets one by one.
- Check the right-hand side of **Status Bar**; you will see something like:

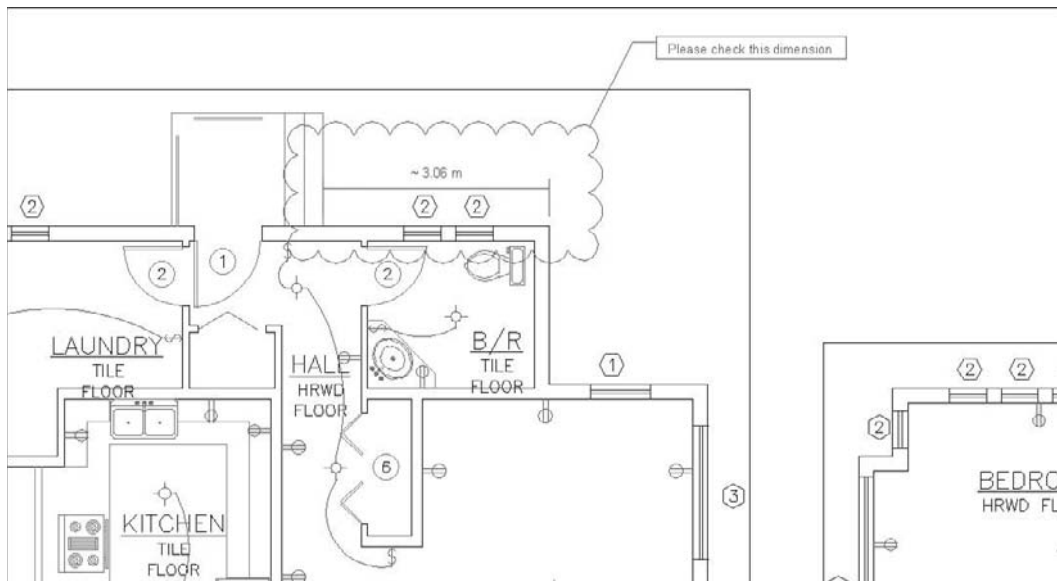


- When done, you will see the following message:



10.21 HOW TO VIEW DWF AND DWFx FILES

- AutoCAD 2010 comes with a free-of-charge viewer called **Autodesk Design Review** to view DWF and DWFx files.
- In order to open a DWF or DWFx file, double-click it and the software will open it immediately.
- In **Autodesk Design Review**, you can make some measurements and add some comments.
- See the following illustration:



CREATING A MULTIPLE-SHEET DWF FILE (METRIC AND IMPERIAL)



Workshops 21-A and 21-B

1. Start AutoCAD 2010.
2. Open the file *Workshop_21.dwg*.

3. Start the **Publish** command.
4. Select the **Model Space** sheet and remove it.
5. Add new sheets by selecting **Add sheets**. Browse for AutoCAD 2010\Sample\Blocks and Tables – Metric.dwg.
6. Remove the **Model Space** sheet.
7. Make the **ISO A1** sheet the top sheet.
8. Select the **Publish to DWF** file.
9. Click the **Publish** button to create the file. It will ask you for the filename and place to save the file in; specify accordingly.
10. **Publish** will be done in the background. You can see this from the tray at the lower right-hand corner of the AutoCAD 2010 window.
11. After the background publishing, browse to the place you saved the file in and double-click it. **Autodesk Design Review** will open automatically and start to show you the contents of the DWF file.

CHAPTER REVIEW

1. What should you control in **Page Setup**?
 - a. Paper size
 - b. Which plotter to send to
 - c. Viewports
 - d. A and B
2. Layouts contain _____ and each can be assigned its own scale.
3. DWF files can be single sheet or multiple sheets.
 - a. True
 - b. False
4. You can choose whether or not to include the Layers in a DWF file.
 - a. True
 - b. False
5. The **Named Plot Style Table** file extension is:
 - a. *filename.ctb*
 - b. *filename.stb*
 - c. *filename.sbt*
 - d. *filename.bct*
6. If you assign a plot style to a layer in a **Named Plot Style Table**, you need to click the _____ button from the **Status bar** to see this lineweight in the layout.

CHAPTER REVIEW ANSWERS

1. d
2. Viewports
3. a
4. a
5. b
6. **Show/Hide Lineweight**

Chapter 11 ADVANCED OBJECTS

In This Chapter

- ◇ Introduction
- ◇ The **Pline** Command: A Revision
- ◇ The **Rectangle** Command
- ◇ The **Polygon** Command
- ◇ The **Donut** Command
- ◇ The **PEDIT** Command: Editing Polylines
- ◇ Polyline Editing Commands
- ◇ The **Construction Line** Command and the **Ray** Command
- ◇ The **Point** Command: An Introduction
- ◇ The **Point** Command: Divide and Measure
- ◇ The **Spline** Command
- ◇ The **Ellipse** Command
- ◇ The **Revision Cloud** Command
- ◇ The **Boundary** Command
- ◇ The **Region** Command: An Introduction
- ◇ The **Region** Command: Boolean Operations

11.1 INTRODUCTION

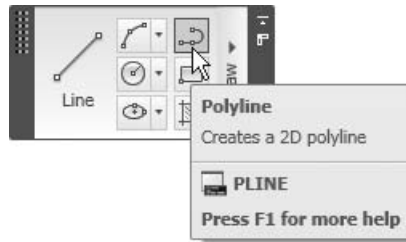
- Earlier, we covered the basic objects used in AutoCAD®, such as the **Line**, **Arc**, **Circle**, and **Pline** commands.
- In this book, we will cover almost all of the other two-dimensional objects used. At the end of this chapter the reader will have a better understanding about how to use the AutoCAD 2D drafting commands.
- Since the **Pline** command is a very important drafting command, it will be covered again here, but with more emphasis on the **PEDIT** command, which is the command used to edit any polyline. Also, we will discuss the other

drafting commands that are based on polylines: the **Rectangle**, **Polygon**, and **Donut** commands. Then we will cover what is special about the polyline in some editing commands, such as the **Fillet** and **Chamfer** commands.

- The **Spline** and **Ellipse** commands are exact curves that help drafters draw real and mathematically correct curved shapes.
- More drafting commands that will help complete our knowledge are the **Point**, **Revision Cloud**, and **Wipeout** commands.
- We will close this chapter by discussing the **Boundary** command. This command will help us create areas out of intersecting arcs. Finally, we will discuss the unique two-dimensional **Region** command.

11.2 THE PLINE COMMAND: A REVISION

- To issue the **Pline** command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Polyline** button.



- The following prompts will appear:

```
Specify start point: (Specify the starting point)
Current line-width is 0.9000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]:
(specify the next point or choose one of the options
available)
```

- After you specify the first point, the **Pline** command will respond with the current polyline width (in our example it is 0.90). You will then be asked to specify the next point. You can use all the methods we learned in the **Line** command.
- If you don't wish to specify the second point, you can choose from the following options:

Arc

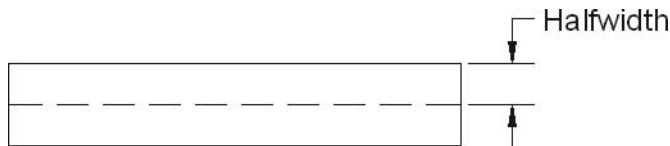
- By default, the **Pline** command will draw lines.
- You can change the mode to draw arcs by selecting the **Arc** option. The following prompt will appear:

Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second
pt/Undo/Width] :

- Earlier we learned that AutoCAD needs three pieces of information to draw an arc.
- AutoCAD already knows the starting point of the arc, which is the starting point of the polyline or the endpoint of the last line segment.
- AutoCAD will make another assumption regarding the direction of the arc, which will be the same angle as the last line segment. The user has the right to accept or reject it.
- If you accept this assumption, then AutoCAD will ask you to specify the endpoint of the arc.
- If you reject this assumption, then you must specify the second piece of information from the following:
 - **Angle** then **Center** or **Radius**
 - **Center** then **Angle** or **Length**
 - **Direction** then **End**
 - **Radius** then **End** or **Angle**
 - **Second** then **End**

Halfwidth

- **Halfwidth** is the first option used to specify the width of a polyline.
- Specify the halfwidth of the polyline from the center to one of its edges. See the following illustration:



- When you select this option, AutoCAD will show the following prompts:

Specify starting half-width <1.0000>:
Specify ending half-width <1.0000>:

- In the preceding example, the halfwidth was 1.0 for both the start and end.

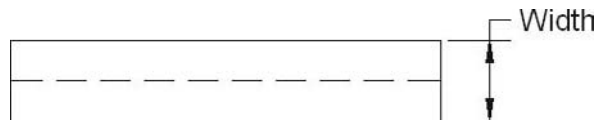
Length

- In the **Pline** command, you should select the **Length** option if you drew an arc, switched to line, and wanted the line to be tangent to the arc.
- This option will assume the angle to be the same as the last segment. Hence, AutoCAD will only ask you for the length, and the following prompt will appear:

Specify length of line:

Width

- The **Width** option is the same as the **Halfwidth** option, but you must input the full width of the polyline. See the following illustration:



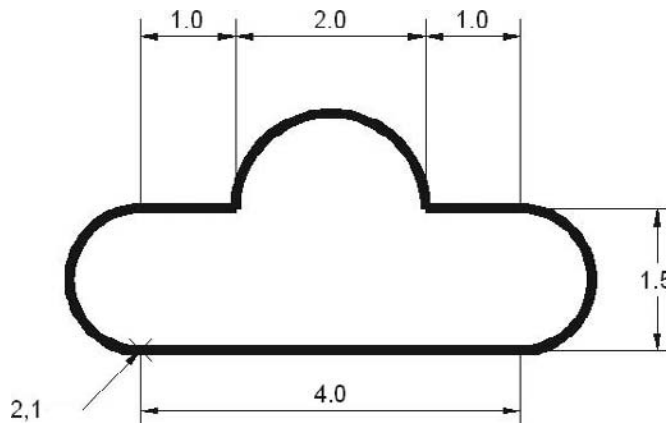
- The **Undo** and **Close** options are the same options as in the **Line** command.
- If you choose to close in the **Arc** option, it will close the shape with an arc.

DRAWING POLYLINES (GENERIC)



Exercise 41

- Start AutoCAD 2010.
- Open the file *Exercise_41.dwg*.
- Using the **ORTHO** command and the **Direct Distance Entry** option, draw the following shape (without dimensions). Use the **Pline** command with **Width** = 0.1.



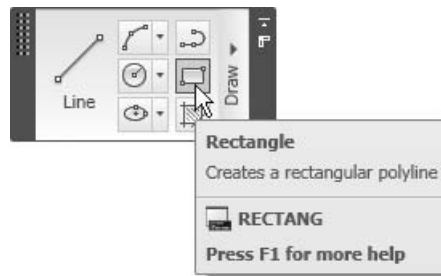
- Save the file and close it.

Hint

- Take note of the following tips:
 - Start from point 2,1 and go to the right.
 - In order to draw the large arc, use **Angle** = 180.
 - Before you draw the last arc, change the mode to **Arc**, and select **Close**.

11.3 THE RECTANGLE COMMAND

- The **Rectangle** command is a **Polyline** command.
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Rectangle** button.



- The following prompt will appear:

```
Specify first corner point or [Chamfer/Elevation/Fillet/
Thickness/Width]: (Specify the first corner)
Specify other corner point or [Area/Dimensions/Rotation]:
(Specify the second corner)
```

- The preceding method is the default method, in which the user will specify two opposite corners.

Chamfer

- You can draw a rectangle with chamfered edges.
- When the first prompt comes up, type **c** and press [Enter]. You will see the following prompts:

```
Specify first chamfer distance for rectangles
<0.0000>: (Specify the first chamfer distance)
Specify second chamfer distance for rectangles
<0.2000>: (Specify the second chamfer distance)
```

- The command then continues normally.

Elevation and Thickness

- These two options are for three-dimensional drawings, not for two-dimensional drawings.

Fillet

- You can draw a rectangle with filleted edges.
- When the first prompt comes up, type **f** and press [Enter]. You will see the following prompt:

Specify fillet radius for rectangles <0.0000>:

- The command then continues normally.

Width

- As with the **Pline** command, you can also draw a rectangle using the **Width** option.
- When the first prompt comes up, type **w** and press [Enter]. You will see the following prompt:

Specify line width for rectangles <0.0000>:

- The command then continues normally.
- After you specify the first corner, the following prompt will appear:

Specify other corner point or [Area/Dimensions/Rotation]:

- The options available are **Area**, **Dimensions**, and **Rotation**.

Area

- Using the **Area** option, you can draw a rectangle with a predefined area value.
- Specify the first corner. When the second prompt appears, type **a** and press [Enter]. You will see the following prompts:

Enter area of rectangle in current units <25.0000>:

(Type in the desired Area value)

Calculate rectangle dimensions based on

[Length/Width] <Length>: **(Select the known dimension, Length is always in x direction)**

Enter rectangle length <10.0000>: **(Type in the value)**

- The rectangle will always be above and to the right of the first corner.

Dimensions

- To use the **Dimensions** option, you can type in the values of the length and the width.
- Specify the first corner. When the second prompt appears, type **d** and press [Enter]. You will see the following prompts:

Specify length for rectangles <10.0000>: (*Input the dimension in X*)

Specify width for rectangles <10.0000>: (*Input the dimension in Y*)

Specify other corner point or [Area/Dimensions/Rotation]:
(*specify where the rectangle will be drawn X (+ve), and Y (-ve), or X (+ve) and Y (+ve), etc.*)

Rotation

- Use the **Rotation** option to draw a rotated rectangle.
- Specify the first corner. When the second prompt appears, type **r** and press [Enter]. You will see the following prompt:

Specify rotation angle or [Pick points] <0>:

- You must then specify the second corner.
- If you changed any or all of the preceding options, the next time you issue the **Rectangle** command, you will see something similar to the following:

Current rectangle modes: Fillet=0.5000 Rotation=45

11.4 THE POLYGON COMMAND

- The **Polygon** command is a **Polyline** command.
- You can use the **Polygon** command to draw an equilateral polygon starting from 3 sides up to 1024 sides.
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Polygon** button.



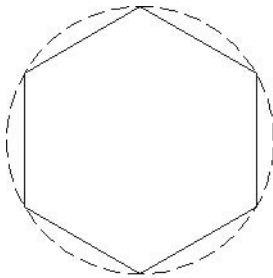
- The following prompt will appear:

Enter number of sides <6>: (*Specify the number of sides*)

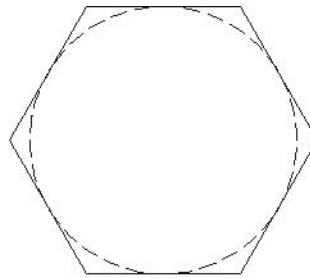
- We now have two options to choose from. Either we know the center of the polygon or we know the length of one of the sides.
- If you know the center, specify it when you see the following prompt:

Specify center of polygon or [Edge]:

- Again, we have two options if we know the center of the polygon. Either the polygon is **Inscribed** inside an imaginary circle, or the polygon is **Circumscribed** about this imaginary circle. See the following illustration:



Inscribed



Circumscribed

- After you specify which option you wish to use, specify the radius of the imaginary circle. The following prompts will appear:

Enter an option [Inscribed in circle/Circumscribed about circle] <I>: (*Specify whether I or C*)

Specify radius of circle: (*Type the value of the Radius*)

- The second option is used if you know the length of one of the sides; accordingly, AutoCAD will draw the other sides. To calculate the length of one of the sides, you must specify two points. This will tell AutoCAD the length and the angle of the first side, thus AutoCAD will know the position of the resulting polygon. The following prompts will appear:

Enter number of sides <4>: (*Specify the number of the sides*)

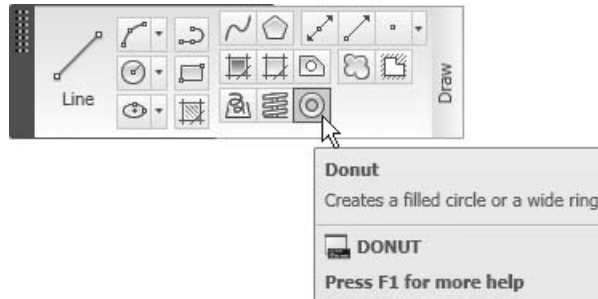
Specify center of polygon or [Edge]: **E**

Specify first endpoint of edge: (*Specify the first point*)

Specify second endpoint of edge: (*Specify the second point*)

11.5 THE DONUT COMMAND

- The **Donut** command is a **Polyline** command.
- Use the **Donut** command to draw a thickened circle by specifying two diameters: inside and outside.
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Donut** button.



- The following prompts will appear:

Specify inside diameter of donut <0.5000>: *(Type the value of the inside diameter)*

Specify outside diameter of donut <1.0000>: *(Type the value of the outside diameter)*

Specify center of donut or <exit>: *(Specify the center for as many times as you wish)*

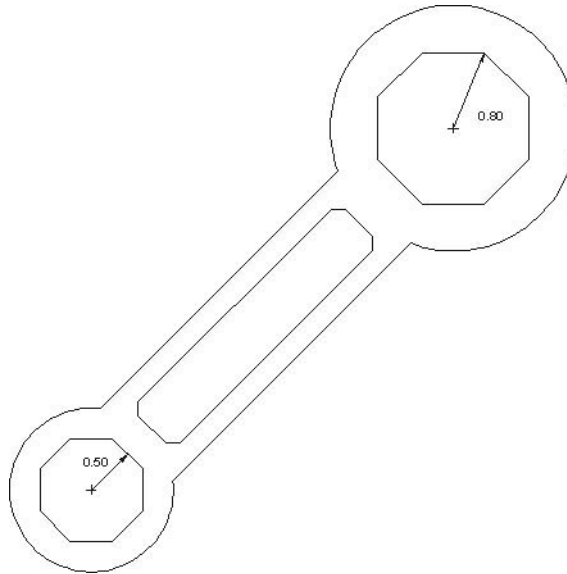
DRAWING RECTANGLES, POLYGONS, AND DONUTS (GENERIC)



Exercise 42

- Start AutoCAD 2010.
- Open the file *Exercise_42.dwg*.
- Draw the two polygons using the information shown in the following illustration.

- Using the two points (turn on the **Node** option in **OSNAP**), draw the rectangle using **Angle** = 45 and **Chamfer** distances = 0.1.



- Save the file and close it.

11.6 THE PEDIT COMMAND: EDITING POLYLINES

- Although polylines respond to the same editing commands as lines do, the **PEDIT** command was created specifically to edit polylines.
- The **PEDIT** command can open a closed polyline, and vice versa.
- It can create curves from straight lines, and vice versa.
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Modify** panel, click the **Edit Polyline** button.



- The following prompts will appear:

Select polyline or [Multiple]: (**Select the desired single polyline to edit**)

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Reverse/Undo]:

Open

- The **Open** option appears only if the selected polyline is closed.
- To open a closed polyline, AutoCAD remembers the last segment and erases it.

Close

- The **Close** option appears only if the selected polyline is open.
- To close an open polyline, AutoCAD remembers the first and the last points and connects them.

Join

- When the first prompt appears, if you select a line or arc for the purpose of joining them together to become a polyline, AutoCAD will issue the following prompt:

Object selected is not a polyline
Do you want to turn it into one? <Y>

- Now the user can select the **Join** option to connect this converted object with all other lines and arcs.

Width

- The **Width** option is used to assign the selected polyline a uniform width for all its segments.
- The following prompt will appear:

Specify new width for all segments: (**Specify the uniform width**)

Edit Vertex

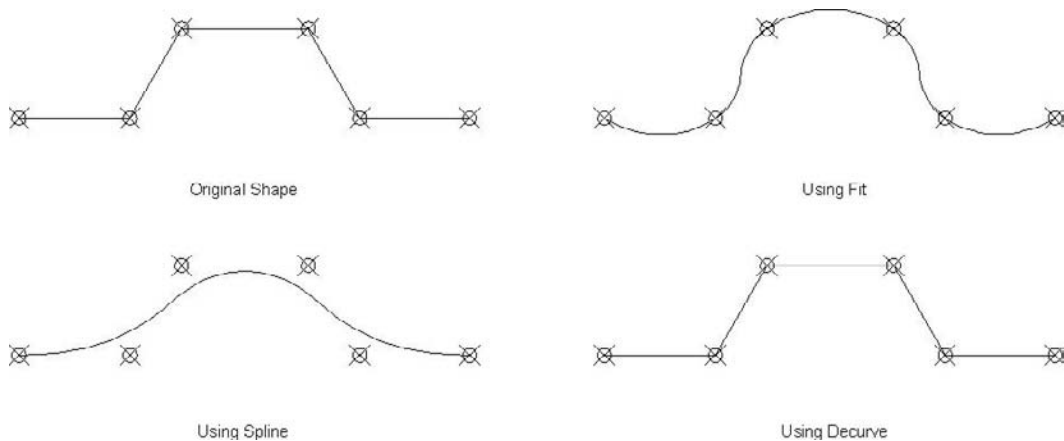
- The **Edit Vertex** option is used to edit the vertices of the selected polyline. It is used to add a new vertex or delete an existing vertex, and so forth.
- This option is very complicated. It is more productive to explode the polyline, make the necessary adjustments, and then join it again, rather than spending unnecessary time trying to find the right option.

- If you choose to use the **Edit Vertex** option, the following prompt will appear:

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/
Width/eXit] <N>:

Fit/Spline/Decurve

- These three options are primarily used to achieve the same results.
- To transfer straight lines of a selected polyline to curves, you can use one of two options:
 - The **Fit** option, which will create an arc connecting the vertices.
 - The **Spline** option, which will use the vertices as controlling points to draw an approximation of a B-spline curve.
- Use the **Decurve** option to return curves to straight segments.
- See the following illustration:

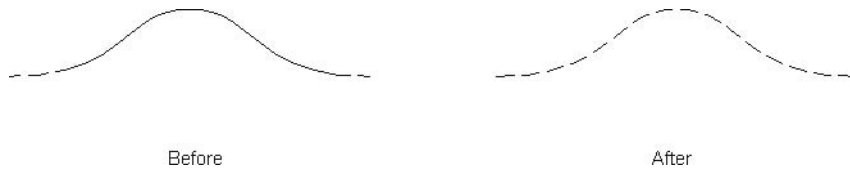


Ltype gen

- If you are converting a straight line segment using the **Spline** option and the lines were dashed, the output of the function will be either a continuous curve or a semi continuous curve. See the following illustration:



- To return to the original specifications, use the **Ltype gen** option. See the following illustration:

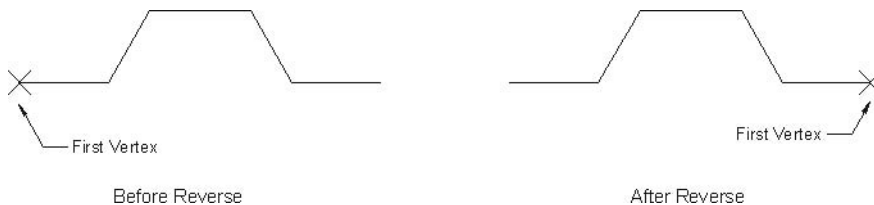


- The following prompt will appear:

Enter polyline linetype generation option [ON/OFF] <Off>: ON
(Type ON if you want to activate it)

Reverse

- The **Reverse** option reverses the order of vertices in a polyline. See the following illustration:



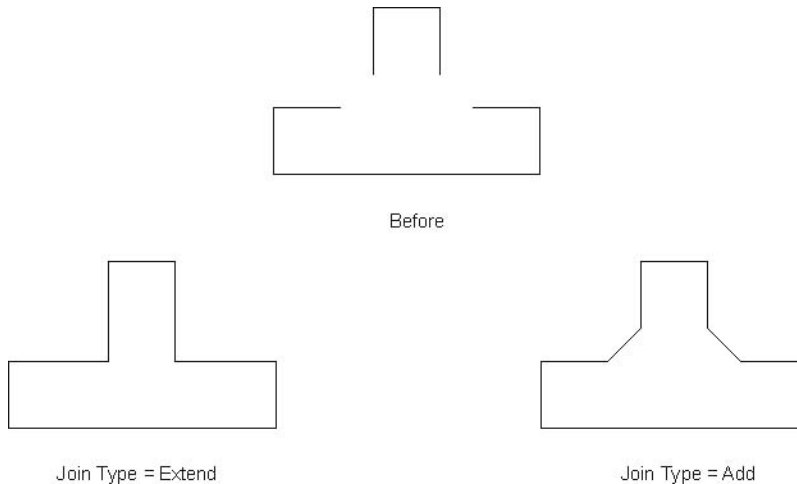
Undo

- Use the **Undo** option to undo the last operation that took place in the **PEDIT** command.

Multiple

- Use the **Multiple** option to edit multiple polylines in one command.
- The options available for single polyline editing are the same for multiple polyline editing.
- One of the advantages of this option is the ability to join two or more polylines together. (Single polyline editing will enable you to join lines and arcs to become polylines. This option will join two or more polylines together to create one object.)
- The user must set two factors in order for the process of joining two polylines to be successful. These factors are:
 - **Fuzz distance** is the maximum distance between the endpoints of the two polylines to be joined. If the actual distance is more than the **Fuzz distance** the joining process will not be completed.
 - **Join Type** will dictate the method of joining, i.e., **Add**, **Extend**, or **Both**.

- The following illustrations demonstrate the difference between **Add** and **Extend**:



- If you are joining lines and arcs to make them polylines and you don't wish to be presented with the following prompt:

Object selected is not a polyline
Do you want to turn it into one? <Y>

then you can use the system variable **PEDITACCEPT** and set it to 1. AutoCAD will make the conversion of the first object selected without asking you each and every time to turn it into one.

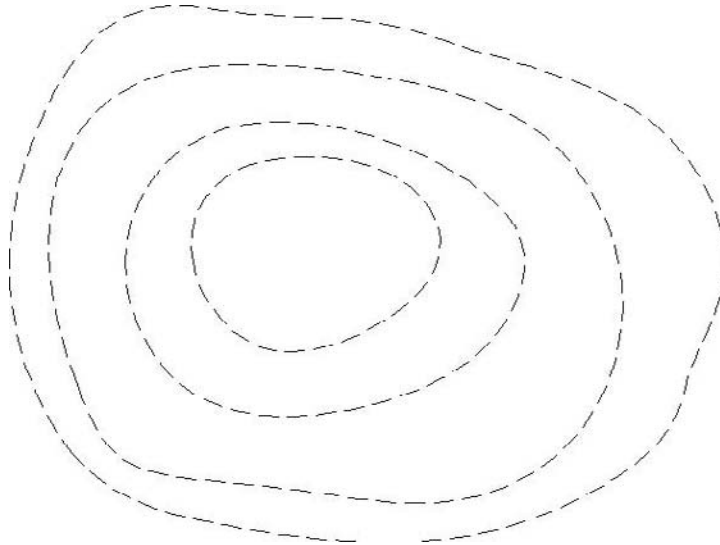
EDITING POLYLINES (GENERIC)



Exercise 43

- Start AutoCAD 2010.
- Open file *Exercise_43.dwg*.
- In the **Command Window**, type **PEDITACCEPT** to make sure that each time you use the **PEDIT** command, AutoCAD will convert lines into polylines automatically and enter the value of 1.
- The following illustration starts with lines. Using the **PEDIT** command, convert each closed shape to a polyline.
- Using the **Multiple** option, convert all polylines to splines.
- Using the **Multiple** option and the **Ltype gen** option, restore the line type of the polylines.

- You should have something like the following:



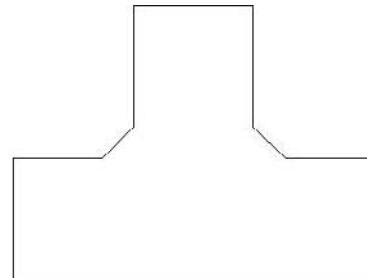
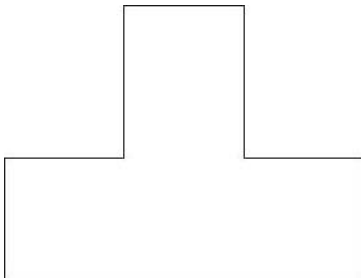
- Save the file and close it.

EDITING POLYLINES (GENERIC)



Exercise 44

- Start AutoCAD 2010.
- Open file *Exercise_44.dwg*.
- Make sure that all of the objects in this file are polylines (click on any object and **Quick Properties** will show that it is a polyline).
- Using the **PEDIT** command and the **Multiple** option, do the following:
 - Using the shapes on the right, join the two polylines as shown (enter a proper **Fuzz Distance** if you know that the length of the openings is 0.5657).
 - Using the shapes on the left, join the two polylines as shown.



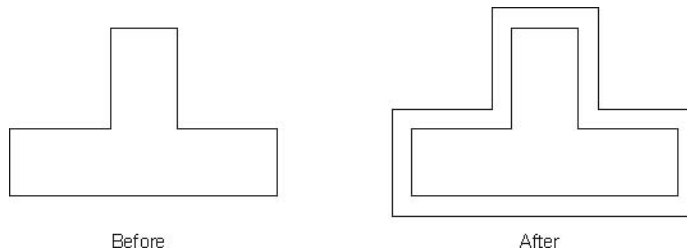
- Save the file and close it.

11.7 POLYLINE EDITING COMMANDS

- Since a polyline is a unique object, some of the editing commands are unique as well.
- These commands are:
 - **Offset**
 - **Fillet** and **Chamfer**
 - **Area**
 - **Quick Properties**
 - **Properties**

Offset

- Since a polyline is one object, when you offset it will be offset as one object.
- Therefore, if you want to draw a plan of a wall, it will be better if you do it using a polyline because it will offset it quickly and easily.
- See the following illustration:

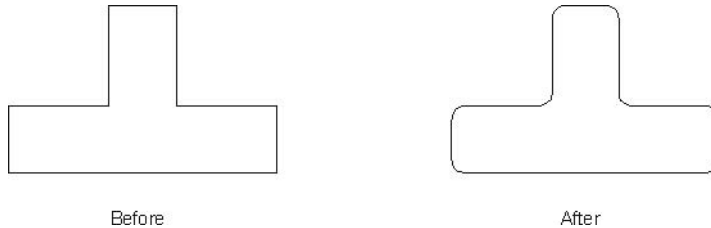


Fillet and Chamfer

- There is a special option in the **Fillet** and **Chamfer** commands used to deal with polylines.
- In the **Fillet** command the following prompts appear:

```
Current settings: Mode = TRIM, Radius = 0.4000
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: P
Select 2D polyline: (Select the desired polyline)
```

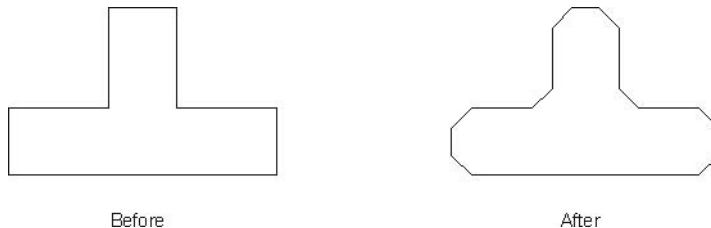
- All of the edges of the plotline will be rounded using the radius specified in one step. See the following illustration:



- In the **Chamfer** command the following prompts appear:

```
(TRIM mode) Current chamfer Dist1 = 0.5000, Dist2 = 0.5000  
Select first line or [Undo/Polyline/Distance/Angle/Trim/  
mEthod/Multiple]: P  
Select 2D polyline: (Select the desired polyline)
```

- All of the edges of the plotline will be chamfered using the distances specified in one step. See the following illustration:



Area

- The **Area** command is a one-step, simple way to get the area of a closed (or even an open) polyline that contains line segments or even lines and arcs.
- Start the **Area** command and type **O** (for object), and select the polyline. The area will be shown right away.

- The prompts will be as follows:

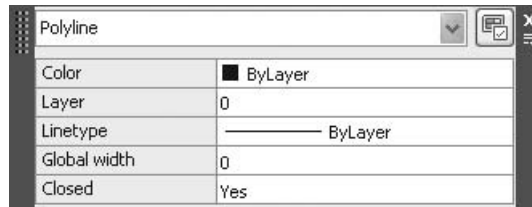
Specify first corner point or [Object/Add/Subtract]: 0

Select objects: (**Select the desired polyline**)

Area = 70.6318, Length = 40.1690

Quick Properties

- Without issuing any commands, select any polyline you would like to edit and the **Quick Properties** window will appear, as seen in the following:

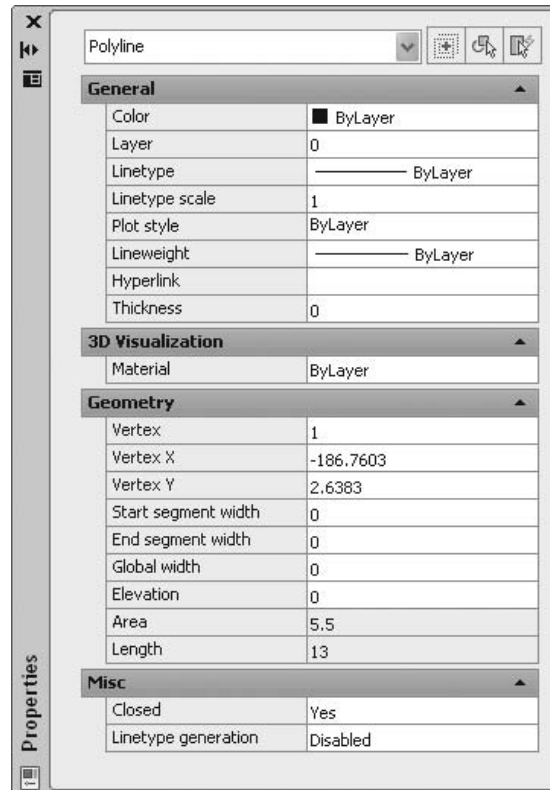


- In this window you can change the following:
 - **Color**
 - **Layer**
 - **Linetype**
 - **Global width**
 - **Closed (Yes/No)**

Properties

- You will find the **Properties** command a very useful tool in editing almost all of the objects in AutoCAD.
- It will not replace the editing commands, but it will make editing objects much easier.

- For instance, if you select a polyline and right-click, and then select from the list **Properties**, you will see something like the following:



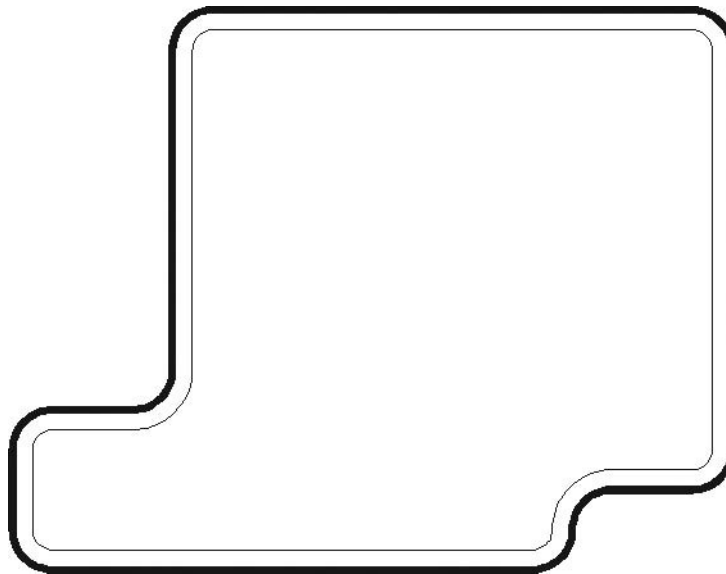
- As you can see from the preceding example, you can do the following:
 - Move through the vertices (**PEDIT/Vertex**).
 - Change the width of the beginning of a segment, the ending of a segment, or the whole polyline.
 - The **Area** and total **Length** are already present without using the **Area** command.
 - Close and open polylines, and vice versa.
 - Make **Linetype** generation just as in the **PEDIT** command.

POLYLINES AND SOME EDITING COMMANDS (IMPERIAL)



Exercise 45

- Start AutoCAD 2010.
- Open file *Exercise_45.dwg*.
- Using the existing polyline, create a fillet with **Radius** = 6".
- Offset the new polyline to the inside using **Offset distance** = 3".
- Using the **Area** command, calculate the inner area. (*Answer: 43.4110 sq. ft.*)
- Using the **Properties** command, what is the total length of the outer polyline?
- Using the **Quick Properties** command, change the **Global Width** to 1".
- You will get the following picture:



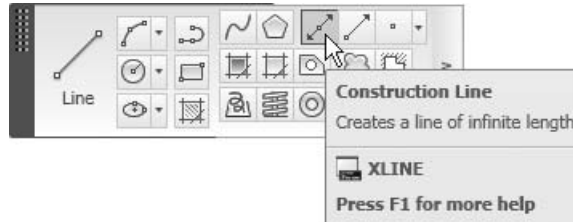
- Save the file and close it.

11.8 THE CONSTRUCTION LINE COMMAND AND THE RAY COMMAND

- The **Construction Line** command will draw an infinite line that extends beyond the screen in two directions using different angle methods.
- The **Ray** command draws a line with a known starting point and an infinite endpoint.
- You can use commands like **Offset**, **Trim**, and **Break** on the two objects.

Construction Line

- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Construction Line** button.



- The following prompt will appear:

Specify a point or [Hor/Ver/Ang/Bisect/Offset]:

- The following sections explain these different options.

Hor

- The **Hor** option is used to draw a horizontal construction line. The following prompt will appear:

Specify through point:

- This prompt will be repeated as many times as you wish. When you are done, press [Enter].

Ver

- The **Ver** option is used to draw a vertical construction line. The following prompt will appear:

Specify through point:

- This prompt will be repeated as many times as you wish. When you are done, press [Enter].

Ang

- The **Ang** option is used to draw a construction line with an angle. The following prompts will appear:

Enter angle of xline (0) or [Reference]: **(Type the desired angle)**

Specify through point:

- First, type in the desired angle and then specify the point that the construction line will go through. This prompt will be repeated as many times as you wish. When you are done, press [Enter].

Bisect

- The **Bisect** option will ask for three points:
 - The construction line will pass through the first point.
 - The second and third points will form an angle; the construction line will bisect this angle.
- The following prompts will appear:

```
Specify angle vertex point:  
Specify angle start point:  
Specify angle end point:
```

- The last prompt will be repeated as many times as you wish. When you are done, press [Enter].

Offset

- If you have an existing line or polyline in your drawing and you want to draw a construction line parallel to it, use the **Offset** option. The following prompts will appear:

```
Specify offset distance or [Through] <Through>:  
Select a line object:  
Specify side to offset:
```

- As you can see, these prompts are identical to the **Offset** command.



- All of the preceding options can be found using the right-click. See the following menu:



Ray

- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Ray** button.



- The following prompts will appear:

Specify start point:

Specify through point:

- The **Ray** command is a very simple command that requires only two points. Specify the starting point and then the second point, which will specify the angle of the ray.
- The last prompt will be repeated as many times as you wish. When you are done, press [Enter].
- If you break a construction line it will become a ray.
- If you trim a construction line from one side, it will become a ray. If you trim it from two sides, it will become a line.
- If you trim a ray, it will become a line.

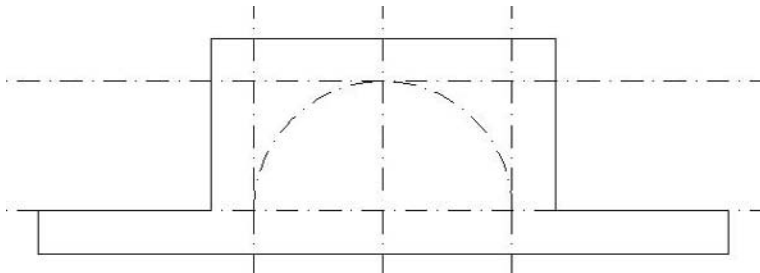


CONSTRUCTION LINE AND RAY COMMANDS (GENERIC)



Exercise 46

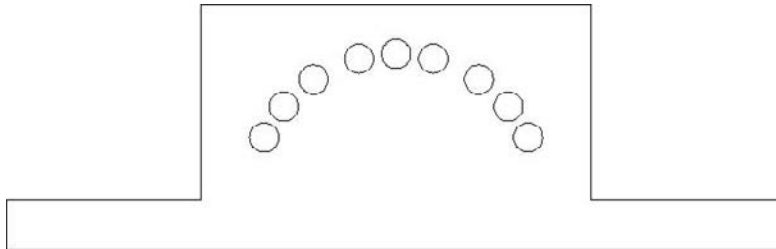
- Start AutoCAD 2010.
- Open the file *Exercise_46.dwg*.
- Make sure you are in the **Construction** layer; if not, make it current.
- Using the **Construction Line** command and the **Offset** option, create two construction lines parallel to the two vertical lines on the inside with **Offset distance** = 1.
- Repeat the preceding step with the upper and lower horizontal lines.
- Using the **Construction Line** command and the **Vertical** option, create one construction line in the center of the top horizontal line.
- Make sure that **Intersection** in **OSNAP** is turned on.
- Draw an arc (one-half of a circle) using the lower three points (**Start/Center/End**).
- You should have the following picture:



- Draw four rays with angles (25, 40, 55, and 75), starting at the center point of the arc.



- Use **Dynamic Input** to get the angles right.
- Make the layer **Openings** the current layer and draw four circles (**D** = 0.5), using the intersection of the rays with the arc, and one circle using the intersection of the middle construction line with the arc.
- Freeze the **Construction** layer.
- Mirror the existing four circles (excluding the one with **Angle** = 90) to the left.
- You should have the following picture:



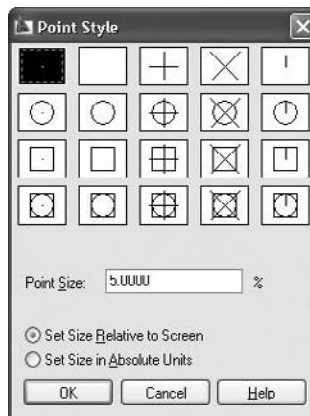
- Save the file and close it.

11.9 THE POINT COMMAND: AN INTRODUCTION

- The **Point** command will help you draw points, either by typing coordinates or by using **OSNAP**.
- The shape of the point will be specified using the **Point Style** command before you insert the point.

Point Style

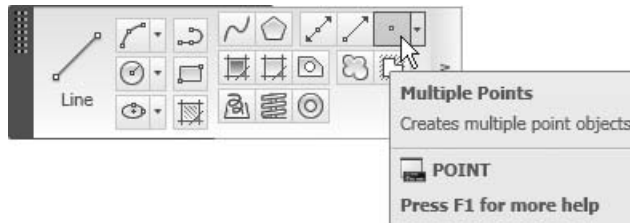
- To issue the **Point Style** command, use the **Command Window** and type **ddptype**.
- The following dialog box will appear:



- In this dialog box you will have 20 different shapes to choose from.
 - Set the size of the point. You have two choices:
 - The size of the point can be relative to the screen size. You must specify a percentage of the screen size.
 - You may also input an absolute value regardless of the screen size.
- NOTE**
- After inserting points in the drawing, you can use the **Point Style** command to change the shape and size, and all the inserted points will change accordingly.
 - There will only be one point style per file.

Point

- The **Point** command is used to place a point in the X,Y,Z space.
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Multiple Points** button.



- The following prompt will appear:

Specify a point:

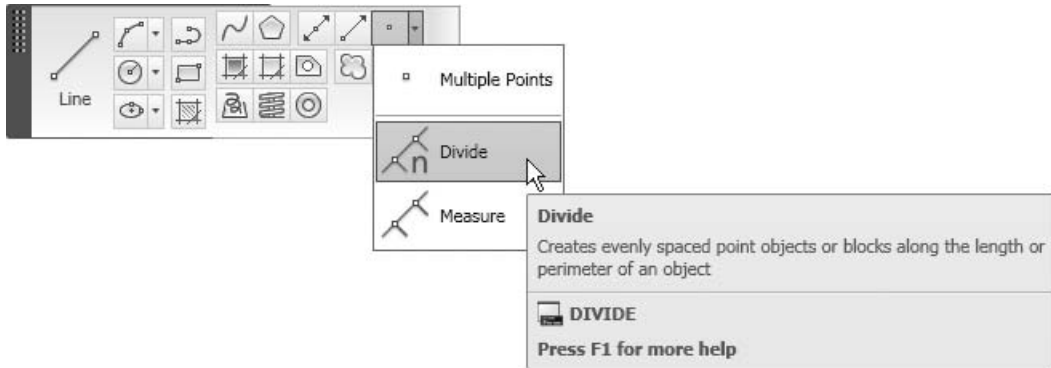
- Click on the desired point. This prompt will be repeated as many times as you wish, and when you are done, press [Esc].
- NOTE**
- After inserting points in the drawing, you can use the **OSNAP NODE** in order to catch the exact point.

11.10 THE POINT COMMAND: DIVIDE AND MEASURE

- Both the **Divide** command and the **Measure** command use points in order to cut objects into segments.
- The **Divide** command uses points to cut any object into equal-length segments specified by the user.
- The **Measure** command uses points to cut any object into lengths specified by the user.

Divide

- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Divide** button.



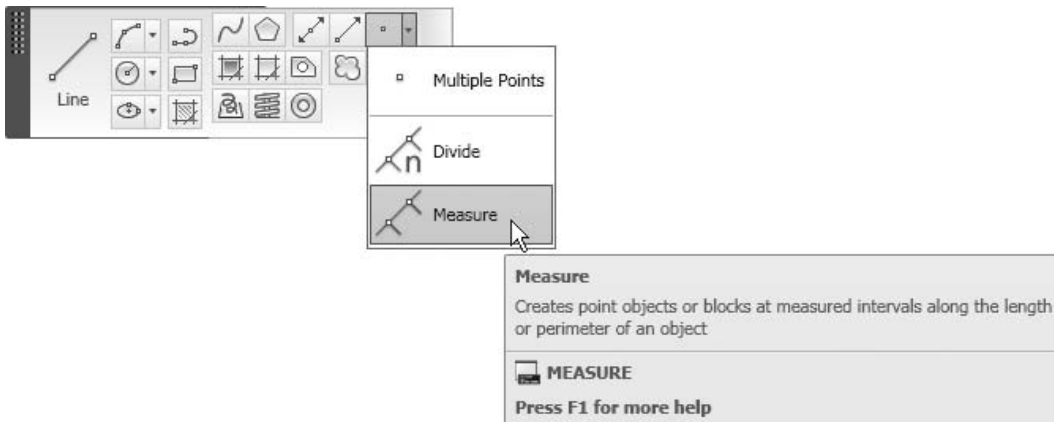
- The following prompts will appear:

Select object to divide: (*Select the desired object*)

Enter the number of segments or [Block]: (*Type in the number of equal spaced segments*)

Measure

- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Measure** button.



- The following prompts will appear:

Select object to measure: (*Select the desired object*)

Specify length of segment or [Block]: (*Type in the length of the segments*)



- It is important to know which end you want to start measuring from; pick the object from the point nearest the desired end.
- AutoCAD will begin inserting points measuring from that end until the left distance is less than the specified distance.



- In both commands you can use **Block** instead of a point. In this case, the following prompts will appear:

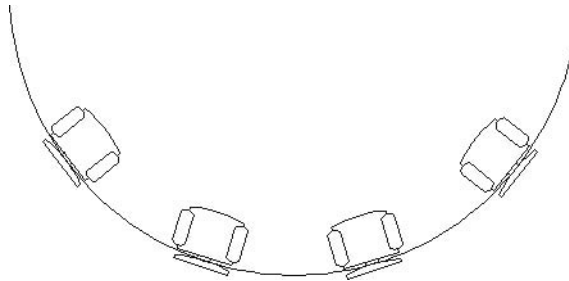
Enter the number of segments or [Block]: **B**

Enter name of block to insert: chair (**Type in the name of the block**)

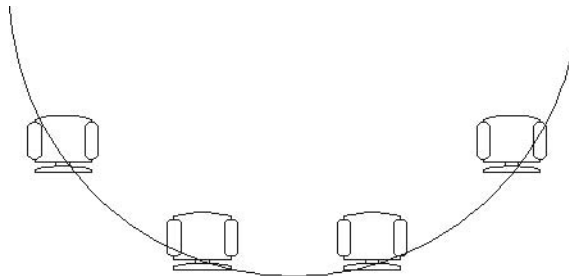
Align block with object? [Yes/No] <Y>: (**Select Yes or No**)

Enter the number of segments: (**Type in the number of segments for Divide and the Length of the segment for Measure**)

- The following illustration demonstrates how to align a block.
 - **Align = Yes**, where the block will move along with the shape of the object, as seen in the following:



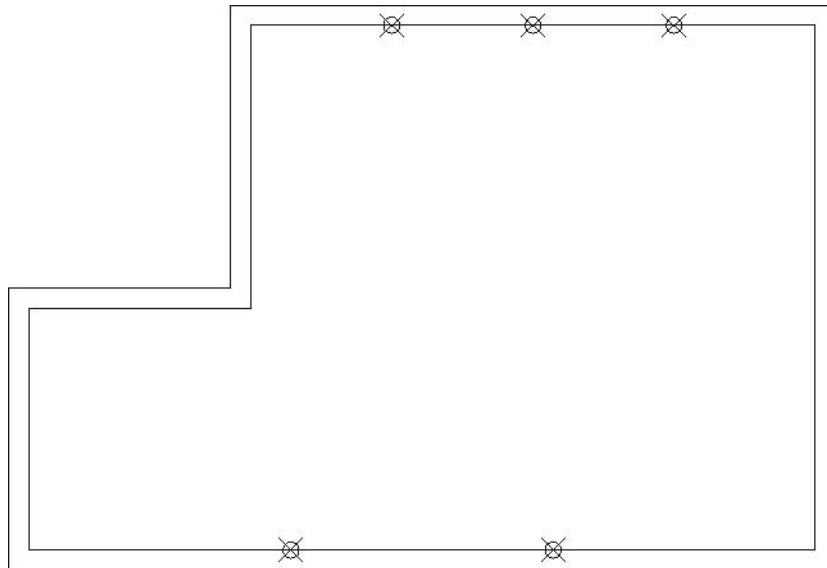
- **Align = No**, where the block will maintain its original angle, as seen in the following:



POINT AND POINT STYLE COMMANDS (IMPERIAL)

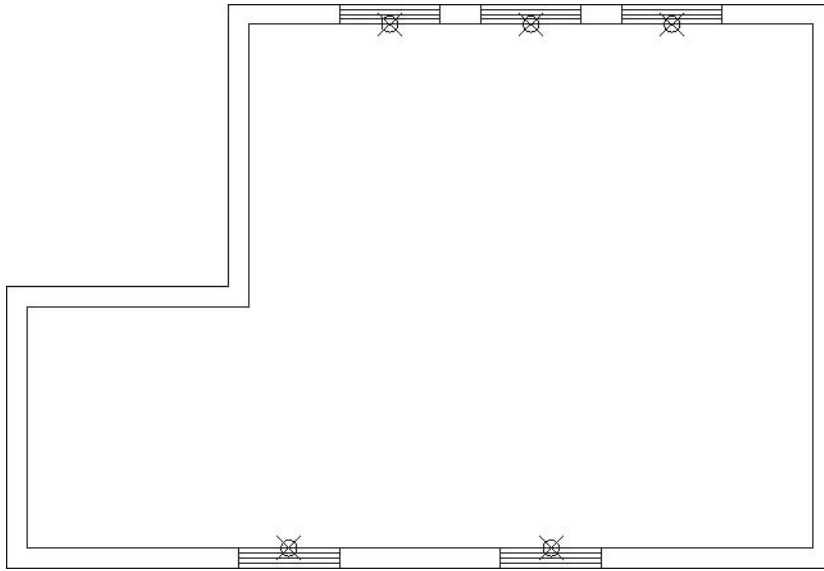
**Exercise 47**

- Start AutoCAD 2010.
- Open file *Exercise_47.dwg*.
- Set the **Point Style** to $\otimes$.
- Explode the inner polyline.
- Using the **Divide** command, divide the upper horizontal line into four equal segments.
- Using the **Measure** command, measure the lower horizontal line with 13' distance, starting from the right of the line.
- Erase the last point at the left of the line.
- You should have the following shape:



- Make sure that **Node** in **OSNAP** is turned on.
- Make the layer **A-Windows** current.
- Insert block **Window**, at the location specified by the points (for the lower ones, use **Angle** = 180).

- You will get the following picture:



- Freeze the **Points** layer.
- Save the file and close it.

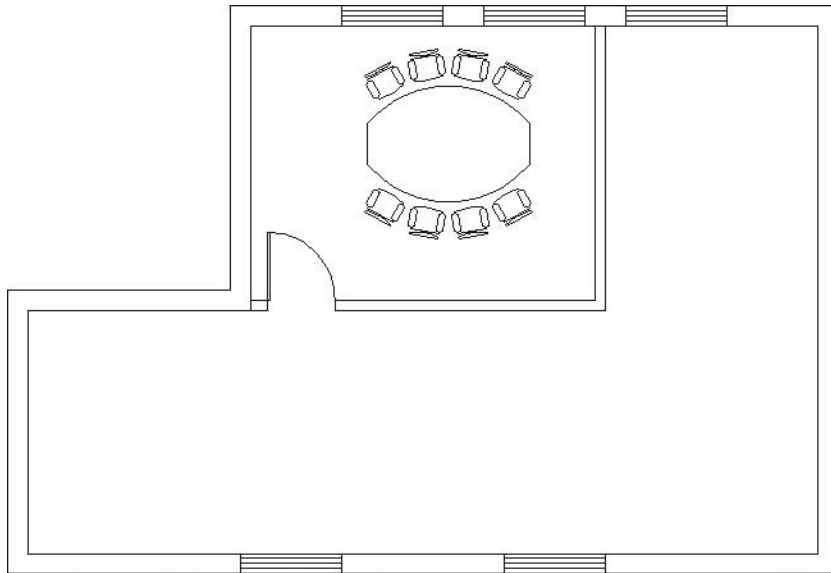
POINT AND POINT STYLE COMMANDS (GENERIC)



Exercise 48

- Start AutoCAD 2010.
- Open the file *Exercise_48.dwg*.
- Divide the two offset arcs using the **Block** command with the name **Chair**.

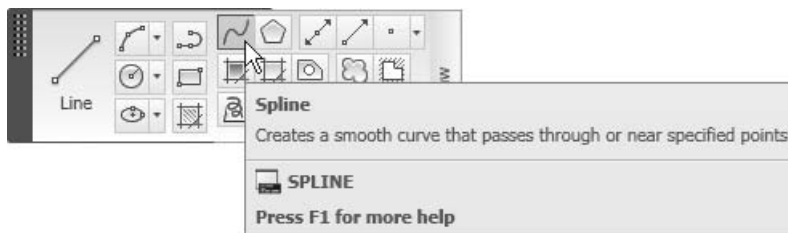
- You can get the rest of the information from the following picture:



- Save the file and close it.

11.11 THE SPLINE COMMAND

- Use the **Spline** command to draw a smooth curve using a series of points based on an exact mathematical method. To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Spline** button.



- The following prompts will appear:

Specify first point or [Object]: (*Specify the first point*)
Specify next point: (*Specify the second point*)
Specify next point or [Close/Fit tolerance] <start tangent>:
(*Specify the third point, or select one of the options*)

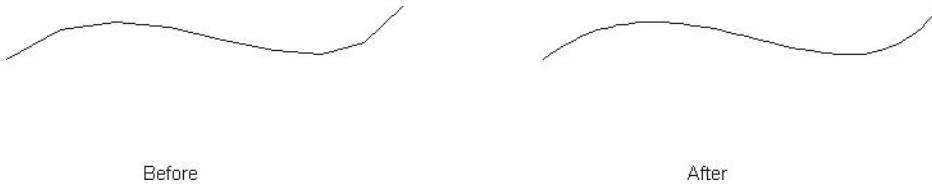
- AutoCAD will draw a curve connecting the points you select. The other options available will help you close the spline the right way.

Object

- If you edit the polyline to be a spline, it will stay as a polyline. You can convert it to be a full spline using the **Object** option.
- The following prompt will appear:

Select objects:

- Select the polyline that has been edited to be a spline and AutoCAD will transform it to be a real spline. See the following illustration:



Close

- In order to use the **Close** option you need to specify an angle of the tangent, which will pass through the starting point.
- The following prompt will appear:

Specify tangent:

Fit Tolerance

- By default, the spline will pass through the specified points. You can choose to make the spline pass near the points, rather than through them, by specifying the **Fit Tolerance** distance.
- The following prompt will appear:

Specify fit tolerance <0.0000>:

Start Tangent

- If you would like to stop after specifying a series of points, but without closing the shape, press [Enter] and the **Start Tangent** option will be selected.
- You must specify two tangents. One that passes through the first point and another that passes through the endpoint.
- The following prompts will be appear:

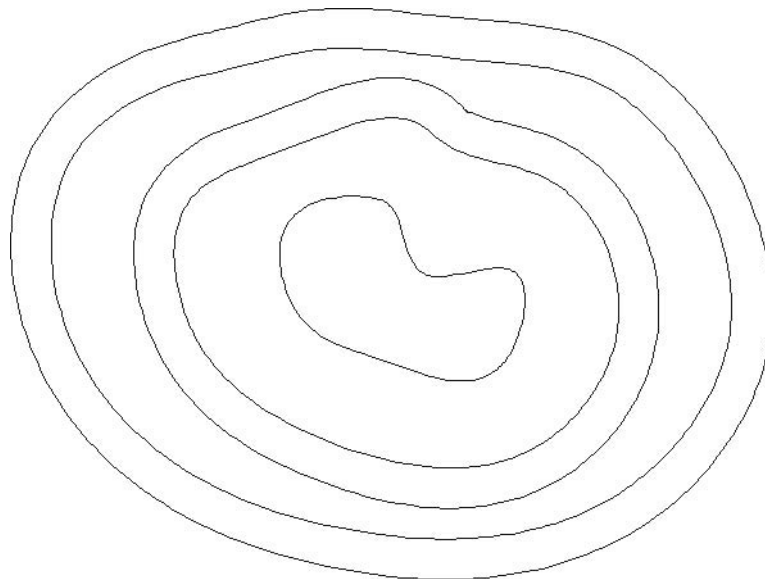
Specify start tangent:

Specify end tangent:

THE SPLINE COMMAND (GENERIC)

**Exercise 49**

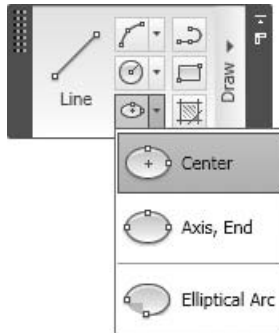
- Start AutoCAD 2010.
- Open the file *Exercise_49.dwg*.
- Using the **Spline** command, draw the inner contour starting from point 1, and continue to the left.
- When you want to close, use **Angle** = 345.
- Freeze layers **Points** and **Text**.
- You should get the following picture:



- Save the file and close it.

11.12 THE ELLIPSE COMMAND

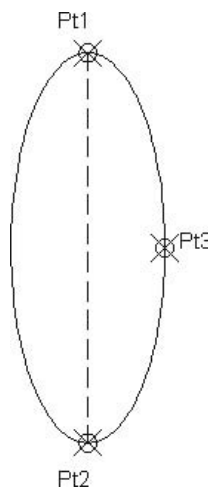
- Use the **Ellipse** command to draw an elliptical shape or an elliptical arc. To issue the command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Ellipse** button.



- There are two ways to draw an ellipse in AutoCAD:
 - **Axis/End**
 - **Center**

Axis/End

- You must specify three points in this method:
 - Two points to specify one of the two axes
 - One point on the other axis
- See the following illustration:



- The following prompts will appear:

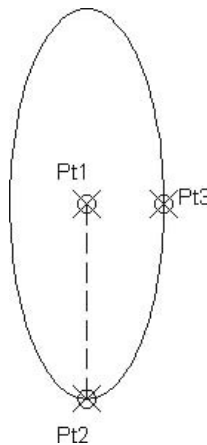
Specify axis endpoint of ellipse or [Arc/Center]:

Specify other endpoint of axis:

Specify distance to other axis or [Rotation]:

Center

- You must specify three points in this method as well:
 - The first point is the center point of the ellipse.
 - The second point is the endpoint of one of the two axes.
 - The third point is the endpoint of the other axis.
- See the following illustration:



- The following prompts will appear:

Specify axis endpoint of ellipse or [Arc/Center]: **C**

Specify center of ellipse:

Specify endpoint of axis:

Specify distance to other axis or [Rotation]:



- Using the two methods, you can use the **Rotation** option as the last piece of information (the third one).
 - When you define the first axis using either method, you are drawing a circle. Imagine that the circle is rotating around this axis.
 - If the rotation angle is equal to 0 or 180, the output will be a circle.
 - If the rotation angle is equal to 90 or 270, the output is nothing.
 - If the rotation angle is equal to any other value, the output is an ellipse.

Elliptical Arc

- To draw an elliptical arc, you must specify the following data:
 - Draw an ellipse using one of the two methods used in the **Ellipse** command.
 - Specify two angles. AutoCAD will draw an elliptical arc counterclockwise between the two angles (it is better to have **Polar Tracking** switched on in order to control the arc graphically rather than by typing angles).
- The following prompts will appear:

Specify axis endpoint of ellipse or [Arc/Center]: **A**
Specify axis endpoint of elliptical arc or [Center]:
Specify other endpoint of axis:
Specify distance to other axis or [Rotation]:
Specify start angle or [Parameter]:
Specify end angle or [Parameter/Included angle]:



- The elliptical arc is not really a separate command because it is part of the **Ellipse** command.

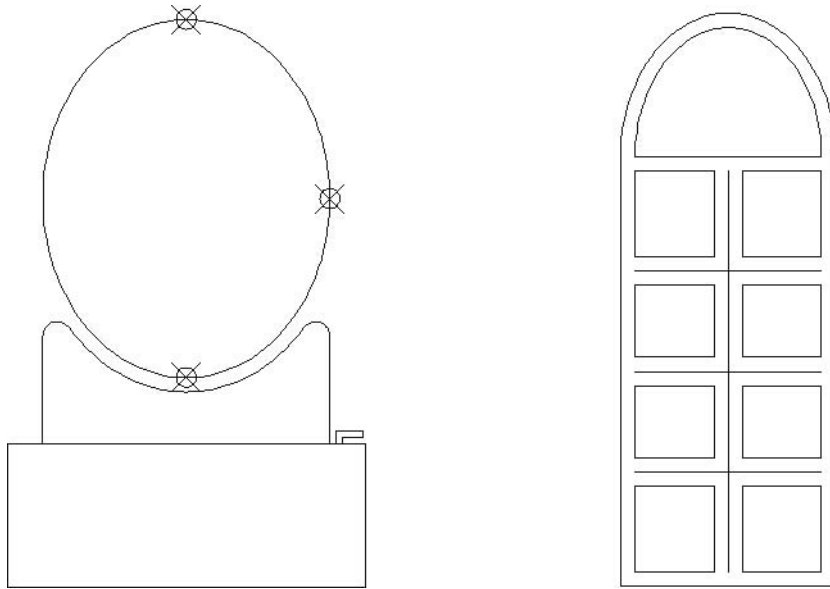
THE ELLIPSE COMMAND (GENERIC)



Exercise 50

- Start AutoCAD 2010.
- Open the file *Exercise_50.dwg*.
- Make sure that **Node** in **OSNAP** is turned on.
- Using the points available, draw the ellipse needed to complete the shape of the toilet.
- Draw an elliptical arc to complete the shape of the window, using the following:
 - The first axis of the ellipse is the width of the window.
 - From the center to the other axis, **Distance** = 2.
 - **Start Angle** = 0, **End Angle** = 180
- Make an offset for the new arc, where **Offset distance** = 0.2 to the inside.

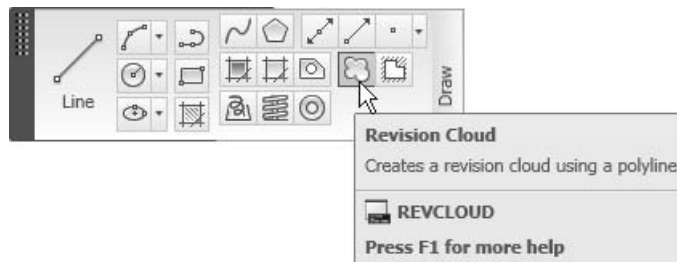
- Make the necessary trimming to create the following shapes:



- Save the file and close it.

11.13 THE REVISION CLOUD COMMAND

- Use the **Revision Cloud** command to add a revision cloud to your drawing.
- This command is a polyline with arcs that connect to each other. All you have to do is click on the desired points and the shape will close automatically.
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Revision Cloud** button.



- The following prompt will appear:

```
Minimum arc length: 15    Maximum arc length: 15    Style:
Normal
Specify start point or [Arc length/Object/Style] <Object>:
```

- Before you start drawing any revision clouds, check the information given to you. The first prompt is telling us the following:
 - **Minimum arc length** = 15
 - **Maximum arc length** = 15
 - **Style** = Normal

Arc Length

- You can change the first two settings using this option. The following prompts will appear:

```
Specify minimum length of arc <15>: 30
Specify maximum length of arc <30>: 50
```

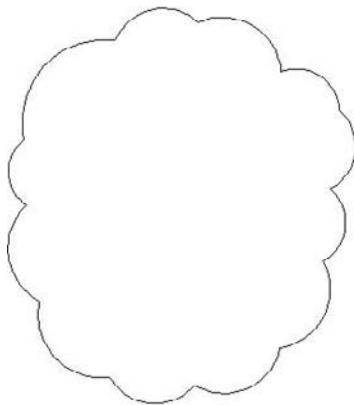
- This means that the lengths of the arcs of the revision clouds will be between 30 and 50.

Style

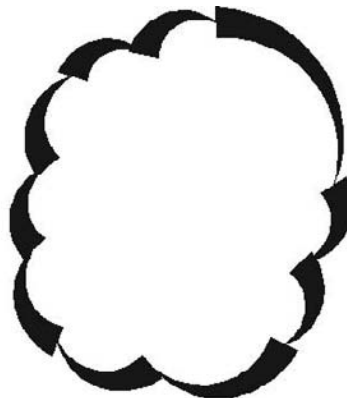
- To change the line style from **Standard** to **Calligraphy**, the following prompt will appear:

```
Specify start point or [Arc length/Object/Style] <Object>: S
Select arc style [Normal/Calligraphy] <Normal>: C
Arc style = Calligraphy
```

- See the following illustration:



Normal



Calligraphy

Object

- To convert any existing object to a revision cloud, the following prompt will appear:

Specify start point or [Arc length/Object/Style] <Object>: **O**

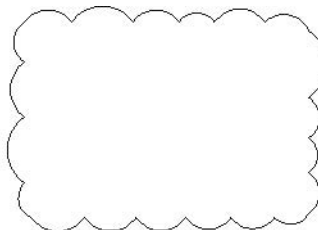
Select object:

Reverse direction [Yes/No] <No>: (*Specify Yes, or No*)

- See the following illustration:



Before



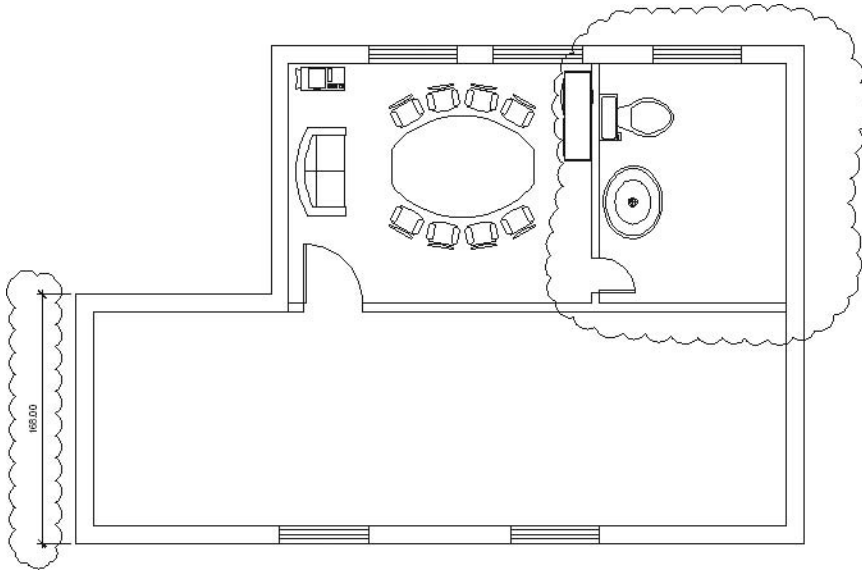
After

THE REVISION CLOUD COMMAND (GENERIC)

**Exercise 51**

- Start AutoCAD 2010.
- Open the file *Exercise_51.dwg*.
- You should notice two things about this drawing:
 - The bathroom furniture is bigger than it should be.
 - The dimension at the left is not correct.
- Do the following two things:
 - Using the **Revision Cloud** command, draw a revision cloud around the bathroom, making the maximum and minimum **Arc Length** = 15.
 - Convert the rectangle around the dimension to a revision cloud.

- You should have something like the following:



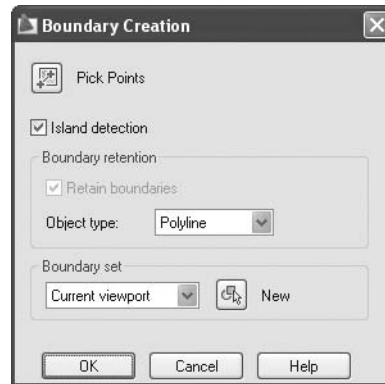
- Save the file and close it.

11.14 THE BOUNDARY COMMAND

- Use the **Boundary** command if you wish to create a polyline (or region—the subject of the next command) from intersecting areas of different objects.
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Boundary** button.



- The following dialog box will appear:



- This button is called **Pick Points**, which is the only way to click inside the area you would like to create a polyline in.

Island Detection

- The **Island detection** option is used to detect any object (like a circle) within the area and consider it as boundary.

Object Type

- Select **Object type** to indicate whether you want to create a polyline or a region (we will discuss regions shortly).

Boundary Set

- The **Boundary set** option provides you with two choices:
 - **Current viewport**: All of the objects in the current viewport will be taken into consideration when AutoCAD creates the polyline.
 - **New**: Only the objects that you will select will be considered when AutoCAD creates the polyline.

THE BOUNDARY COMMAND (GENERIC)



Exercise 52

- Start AutoCAD 2010.
- Open the file *Exercise_52.dwg*.
- Take a good look at the drawing in the file. Make sure you understand where the cut is and where the fill areas are.

- Freeze layer **Hatch**.
- Using the existing and proposed shapes of the road, calculate the cut and fill areas using the **Boundary** command to create a polyline. Then, move the polyline and check the **Properties** (or use the **Area** command and the **Object** option).
- What is the total cut area? _____
(Answer: 18.4733)
- What is the total fill area? _____
(Answer: $3.2424 + 4.8494 = 8.0918$)
- Save the file and close it.

11.15 THE REGION COMMAND: AN INTRODUCTION

- All of the two-dimensional objects we learned in the past are considered *wireframes*. A region is the only two-dimensional object that is considered to be different.
- A region is a very *thin-shelled* object. Hence, any holes inside the region are considered part of the object and not as an isolated object (as they would be in polylines).
- To issue this command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Draw** panel, click the **Region** button.



- There are two ways to create a region in AutoCAD:
 - The **Region** command
 - The **Boundary** command

Region

- The **Region** command will convert any two-dimensional *closed* shape to a region. For example, this command will convert any combination of lines, arcs, polylines, splines, or any closed shapes like a circle or an ellipse.

- The following prompt will appear:

Select objects:

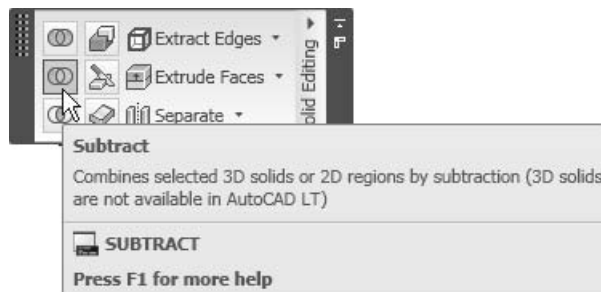
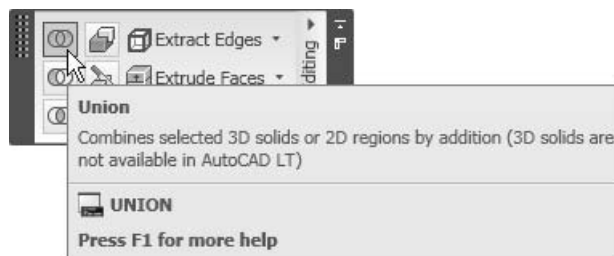
- Select the objects that will formulate a closed shape. Once you are done, press [Enter]. The selected objects will be transformed to regions.

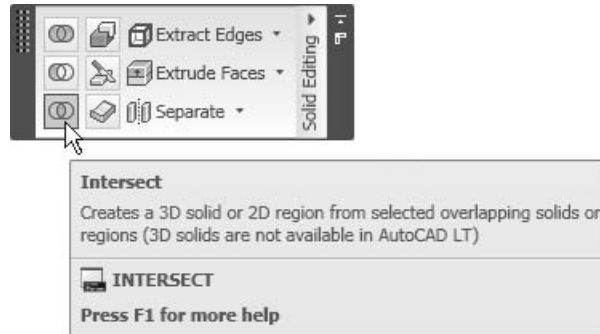
Boundary

- Using the **Boundary** command, select **Object type** to be **Region**.
- Once you have clicked inside the desired area, the object created will be a region.

11.16 THE REGION COMMAND: BOOLEAN OPERATIONS

- Region objects are the only two-dimensional objects that will accept **Boolean Operation** commands. These command are:
 - **Union**, which is used to combine two or more regions to create a new single region.
 - **Subtract**, which is used to subtract a group of regions from another group of regions.
 - **Intersect**, which is used to produce an intersecting area from a group of regions.
- To reach these three commands (you have to change the workspace to 3D Modeling), make sure you are on the **Home** tab on the **Ribbon** and, using the **Solid Editing** panel, select one of the three commands:





- The **Union** command and the **Intersect** command both have the same prompt:

Select Objects:

- You can select the objects in any order.
- When using the **Subtract** command, you will see the following prompts:

Select solids and regions to subtract from...

Select objects:

Select objects: Select solids and regions to subtract...

Select objects:

- First, select the region(s) you want to subtract from and press [Enter]. Then, select the region(s) to be subtracted.
- One of the unique things about regions is that you can get physical information about your two-dimensional object.
- To issue this command, use **Command Window** and type **massprop**.
- You will get something like the following listing:

```
Area:                        803004.1226
Perimeter:                   5347.2674
Bounding box:                X: 1360.1537  --  2511.5395
                             Y: 324.3740   --  1475.7599
Centroid:                    X: 1935.8466
                             Y: 900.0670
Moments of inertia:          X: 7.3228E+11
                             Y: 3.0910E+12
Product of inertia:          XY: 1.3991E+12
Radii of gyration:           X: 954.9506
                             Y: 1961.9664
```


Principal moments and X-Y directions about centroid:

I: 81753968390.5518 along

[1.0000 0.0000]

J: 81753968390.5518 along

[0.0000 1.0000]

THE REGION COMMAND (IMPERIAL)



Exercise 53

- Start AutoCAD 2010.
- Open the file *Exercise_53.dwg*.
- Using the **Boundary** command, convert all shapes to regions.
- Using the **Union** command, unite the upper two circles and the rectangle.
- Repeat the last step for the lower circles.
- Using the **Subtract** command, subtract all the shapes from the outer shape.
- Using **Properties**, what is the net area of the shape? _____
(Answer: 22.4 sq. in.)
- Save the file and close it.

CHAPTER REVIEW

1. The object type that is different from all other two-dimensional objects is:
 - a. **Boundary**
 - b. **Polyline**
 - c. **Region**
 - d. All of the above
2. **Axis/End** is one method used to draw _____.
3. If you convert a polyline from line segments to curves, you may lose the **Linetype** of the polyline.
 - a. True
 - b. False
4. The **Island Detection** option in the **Boundary** command allows the user to pick the objects that will be taken into consideration when AutoCAD creates a polyline.
 - a. True
 - b. False

5. When using the **Divide** and **Measure** commands, you can:
 - a. Distribute points along a selected object
 - b. Distribute blocks along a selected object
 - c. Align blocks over a selected object
 - d. All of the above
6. In the **Fillet** and **Chamfer** commands, there is a special option for _____.

CHAPTER REVIEW ANSWERS

1. c
2. an ellipse
3. a
4. b
5. d
6. polylines

Chapter 12 ADVANCED TECHNIQUES

In This Chapter

- ◇ The **Offset** Command: Advanced Options
- ◇ The **Trim** and **Extend** Commands: Advanced Options
- ◇ Opening More Than One AutoCAD® File
- ◇ Importing Objects from Other Applications
- ◇ The **Hyperlink** Command
- ◇ The **View** Command
- ◇ Using Fields
- ◇ Formulas and Tables
- ◇ Advanced Table Features
- ◇ The **Quick Select** Command
- ◇ The Partial Open and Partial Load Features
- ◇ The Annotative Feature

12.1 THE OFFSET COMMAND: ADVANCED OPTIONS

- Many AutoCAD users are using default options with the **Offset**, **Trim**, and **Extend** commands without looking at the other options—options that would help them do their job faster and better.
- In this section of Chapter 12 we will focus on the options of these three commands that were overlooked.
- When you start the **Offset** command you will see the following prompt:

```
Current settings: Erase source=No Layer=Source  
OFFSETGAPTYPE=0  
Specify offset distance or [Through/Erase/Layer] <Through>:
```

- The first line will state:
 - **Erase source** = No
 - **Layer** = Source
 - **OFFSETGAPTYPE** = 0
- In the second line you will see the following options:
 - Erase
 - Layer

Erase

- If you select the **Erase** option, the following prompt will appear:

Erase source object after offsetting? [Yes/No] <No>:

- If you select **No**, then your original object will remain intact, along with the parallel object generated.
- If you select **Yes**, then your original object will be deleted, while the parallel object generated will remain.

Layer

- By default, the offset object will reside in the same layer that the original object resides in, regardless of the current layer.
- Using the **Layer** option, you can elect to send the duplicate object to the current layer and not to the original object's layer.
- The following prompt will appear:

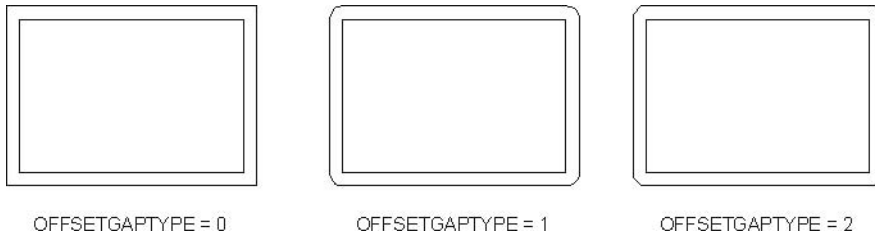
Enter layer option for offset objects [Current/Source]
<Source>:

- As you can see, the default option is **Source**, but you have the choice to input **Current**.

OFFSETGAPTYPE

- This is a **System Variable**, which controls how AutoCAD will treat polylines when they are offset to bigger polylines. It also controls how the gaps generated from the offset will be treated.
- There are three options:
 - **OFFSETGAPTYPE** = 0, which means the lines will be extended.
 - **OFFSETGAPTYPE** = 1, which means the angles will be filleted.
 - **OFFSETGAPTYPE** = 2, which means the angles will be chamfered.

- See the following illustration:



- These settings will affect all of the files, so be careful.

ADVANCED OPTIONS IN OFFSET (IMPERIAL)



Exercise 54

- Start AutoCAD 2010.
- Open the file *Exercise_54.dwg*.
- The shape was drawn in layer 0 (zero).
- Using the **Command Window**, type **OFFSETGAPTYPE** and set the value to 1.
- Start the **Offset** command with the following settings:
 - **Layer** = Current
 - **Erase** = Yes
- Offset the shape to the outside using **Offset Distance** = 1'.
- What is the area of the new polyline? _____ (Answer: 2019.1094 sq. ft.).
- Save the file and close it.

12.2 THE TRIM AND EXTEND COMMANDS: ADVANCED OPTIONS

- We covered almost all of **Trim** and **Extend** options earlier. Two options remain to be discussed:
 - **Project** (applicable in three-dimensional drawings only)
 - **Edge**

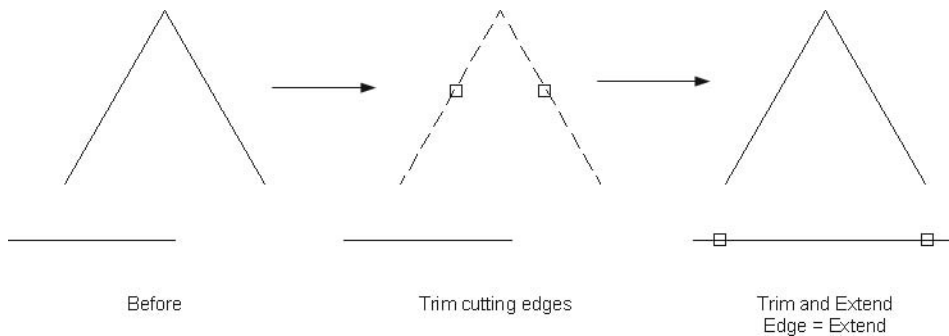
Edge

- By default, in order for the **Trim** command to be successful, the cutting edge(s) and the objects to be trimmed should intersect.

- Also by default, for the **Extend** command to be successful, the boundary edge(s) and objects to be extended should have proposed intersecting points.
- The **Edge** option can trim objects based on extrapolated cutting edges, and extend objects based on extrapolated boundary edges.
- If you select the **Edge** option, you will see the following prompt:

Enter an implied edge extension mode [Extend/No extend]
<No extend>: **E**

- By default, the mode is set to **No extend**. Select mode **Extend** by typing **E** or by right-clicking and selecting the **Extend** option.
- See the following example:



- In the preceding example, we used the **Trim** command and the [Shift] key to change the mode from **Trim** to **Extend**.
- These settings will affect all of the files.

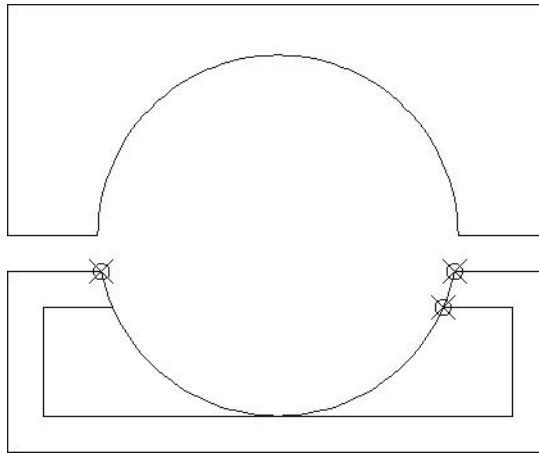
ADVANCED OPTIONS IN THE TRIM AND EXTEND COMMANDS (GENERIC)



Exercise 55

- Start AutoCAD 2010.
- Open the file *Exercise_55.dwg*.
- Start the **Trim** command and select the arc as the cutting edge.
- Set the **Edge** to **Extend**.

- Trim the upper two horizontal lines.
- Extend the lower two horizontal lines using the [Shift] key to flip the command to **Extend**.
- Draw an arc using the **3-Point** method with the following points:
 - **Point 1** = the endpoint of the upper horizontal line at the right.
 - **Point 2** = the endpoint of the lower horizontal line at the right.
 - **Point 3** = the endpoint of the upper horizontal line at the left.
- You should have the following picture:

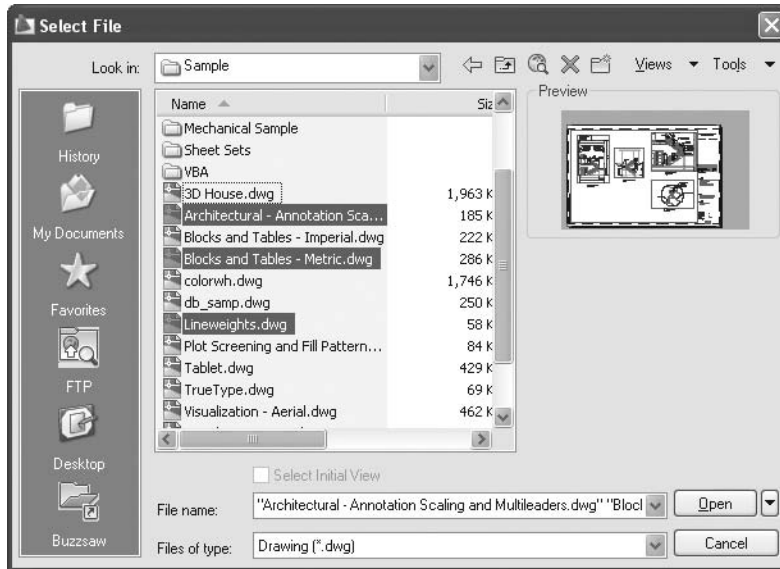


- Save the file and close it.

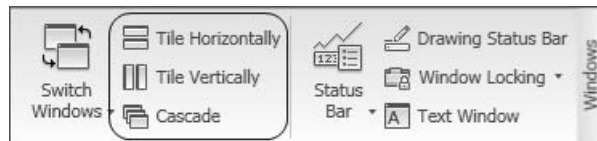
12.3 OPENING MORE THAN ONE AUTOCAD FILE

- Earlier we covered the **Design Center**, which allows us to share **Blocks**, **Layers**, **Text Styles**, and so on, between files. We did not cover sharing objects between files.
- Since AutoCAD allows us to open more than one file, why not take advantage of this feature to copy and paste objects between files?

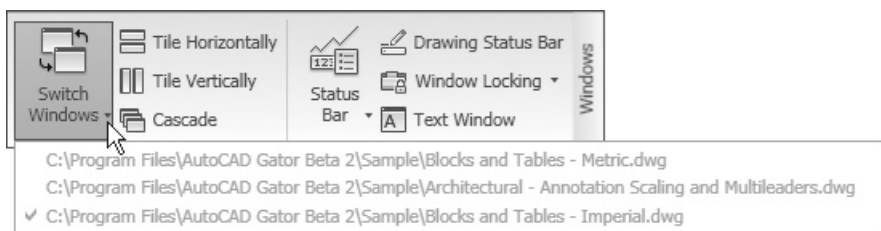
- Using the **Open** command, you can select more than one file at the same time using the [Ctrl] key (you can also do this one-by-one). See the following illustration:



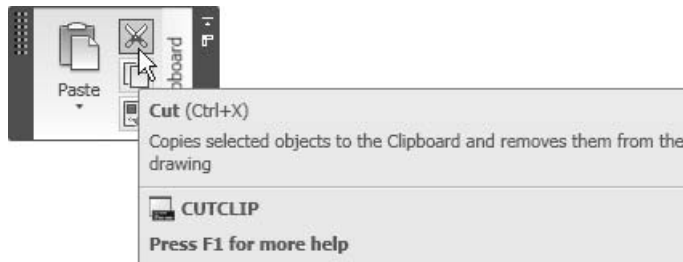
- When you open more than one file, you can use the **View** tab and the **Window** panel to control these files:



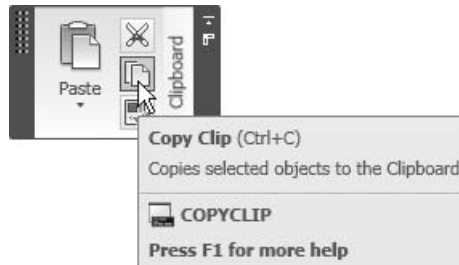
- All of these commands are common to all Windows® applications.
- You can see a list of the opened files using the **Switch Windows** button:



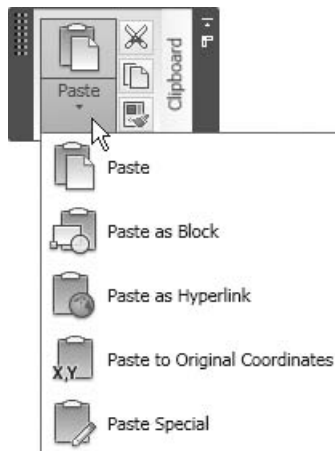
- We have to differentiate between two AutoCAD commands:
 - **Modify/Copy** will copy objects within a single file.
 - **Edit/Copy** will copy objects from one file to another file.
- On the **Home** tab on the **Ribbon** and, using the **Clipboard** panel, you will see a **Cut** command, a **Copy** command, and five types of **Paste** commands.
- The **Cut** command window appears as follows:



- The **Copy** command window appears as follows:



- The **Paste** commands window appears as follows:





- There are two other commands for **Copy**. You can issue them by typing **copybase** or **copylink**.
- The following explains each of the eight commands.

Copy

- The **Copy** command is used to copy selected objects from one file to another file without specifying a base point.

Copy with Base Point

- **Copy with Base Point** is used to copy selected objects from one file to another file by specifying a base point.
- Type **copybase** in the **Command Window**.
- A prompt will appear asking you to specify a base point.

Specify base point:

Copy Link

- The **Copy Link** command is used to copy objects in the current view without selecting an object or specifying a base point.

Paste

- The **Paste** command is used to paste objects in the current drawing.

Paste as Block

- Although you selected objects, you can paste these objects in the current file as blocks. AutoCAD assigns a random name. You can use the **Rename** command to assign a desired name.

Paste as Hyperlink

- This command is only applicable when copying objects from other applications to AutoCAD.

Paste to Original Coordinates

- This command is used to paste selected objects using the same coordinates as the source file.

Paste Special

- The **Paste Special** command is also used for objects copied from other applications (we will discuss it shortly).



- We used the **Match Properties** command earlier. You can use this command across files.
- If you matched an object that resides in a layer with an object in another file in which this layer doesn't exist, AutoCAD will create the same layer in the destination file.
- The same applies for all types of styles: **Dimension**, **Text**, **Multileader**, **Table**, and so on.



- You can select objects and drag-and-drop them (avoiding grips) to other files using the left button or right button.

OPENING MORE THAN ONE AUTOCAD FILE (GENERIC)



Exercise 56

- Start AutoCAD 2010.
- Open the files *Exercise_56_First.dwg* and *Exercise_56_Second.dwg*.
- In *Exercise_56_Second.dwg*, check the following:
 - Is there a layer called "Furniture"? _____
- Tile both files vertically.
- In *Exercise_56_First.dwg*, copy the table and chairs. Specify a base point using the upper left corner of the room.
- In *Exercise_56_Second.dwg*, paste the objects as a block.
- Type **Rename** in the **Command Window**. You will find a block with a random name—something like A\$C4A1D2F06. Rename the block "Table and Chairs."
- Again, check the following:
 - Is there a layer called "Furniture"? _____
- Save the files and close them.

OPENING MORE THAN ONE AUTOCAD FILE (GENERIC)



Exercise 57

- Start AutoCAD 2010.
- Open files *Exercise_57_First.dwg* and *Exercise_57_Second.dwg*.
- Tile the two files vertically.
- Check *Exercise_57_Second.dwg* for the following:
 - Is there a layer called "text"? _____
 - Is there a **Text Style** called "Georgia_09"? _____
- The text in *Exercise_57_Second* is written in Layer 0 (zero).
- Make *Exercise_57_First* the current file.

- Using the **Match Properties** command, select the text as the **Source** object.
- Go to *Exercise_57_Second* and select the text as **Destination**.
- Check *Exercise_57_Second.dwg* again:
 - Is there a layer called “text”? _____
 - Is there a **Text Style** called “Georgia_09”? _____
- Save the files and close them.

12.4 IMPORTING OBJECTS FROM OTHER APPLICATIONS

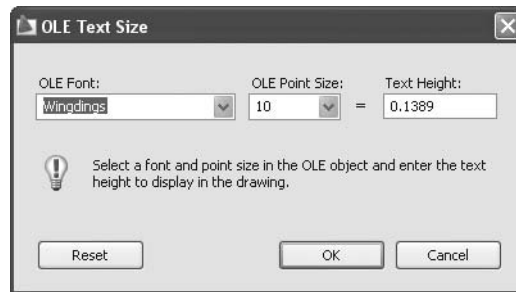
- “Instead of writing the text in AutoCAD, I will let [someone] type up the notes in Microsoft® Word® and I will import it to my AutoCAD drawing,” one engineer said.
- “Instead of creating tables in AutoCAD, I will ask one of the computer wizards in my company to create them in Microsoft Excel® and I will paste them in my drawing,” another engineer claimed.
- Both of these engineers are 100% correct. You can import any object from any application into AutoCAD, and vice versa.
- **OLE** (Object Linking and Embedding) is a Windows feature and all applications that work within Windows can utilize it.
- Linking means there is a *connection* between the source application data and the image in the destination application; therefore, if any change takes place in the original file, the image *will reflect* this change.
- Embedding means there is *no connection* between the source application data and the image in the destination application; therefore, if any change takes place in the original file, the image *will not reflect* this change.
- We will use the same commands we used in the previous pages: **Cut**, **Copy**, and **Paste**.
- If you want to copy objects from other applications to AutoCAD, perform the following steps:
 - Start the desired application and open the desired file.
 - Select the object(s) to be copied.
 - If you want to copy and delete them, use **Edit/Cut**. If you want to keep the original object(s), use **Edit/Copy**.
 - Start AutoCAD and open the file in which you would like to import the object(s).
 - You can use the **Paste** command if you don’t want any link to be established. If you wish to establish a link, you can use the **Paste Special** command.

Edit/Paste

- When you use the **Paste** command, the following prompt will appear:

Specify insertion point: (*Specify an insertion point*)

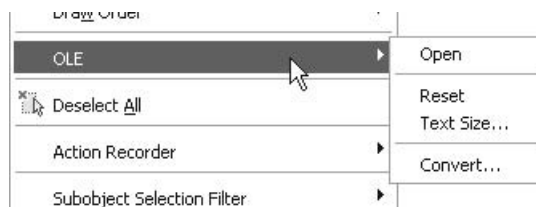
- Subsequently, the following dialog box will appear:



- The dialog box will list the font and font size used in the Windows application. You must specify the AutoCAD text height. Once you are done, click **OK**.
- If the pasted object is in simple **ASCII** format (text coming from a Notepad application or something similar), the object will be pasted as an **MTEXT** object. Other types of objects will be pasted as **OLE** objects.

OLE Editing

- In order to edit an **OLE** object, select it and right-click. The following menu will appear:



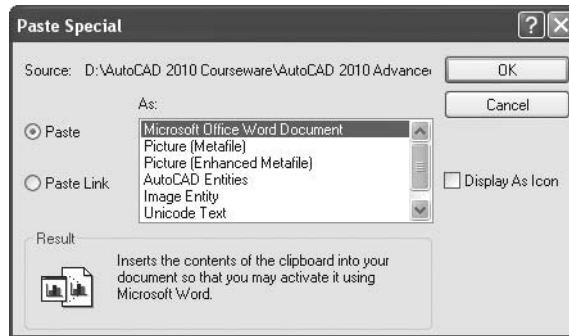
- If you want to open the source application, double-click on the **OLE** object or use right-click and select the **OLE/Open** option.
- Select the **OLE/Reset** option to change the font and font size to the original source font and font size.
- Select the **OLE/Text Size** option to change the text size to a new value (just like the dialog box that appears with the **Edit/Paste** command).
- Select the **OLE/Convert** option to change the nature of the OLE object.

Edit/Paste Special

- When using the **Paste Special** command you will have to choose between two options:
 - **Paste**
 - **Paste Link**

Paste Option

- The **Paste** option allows you to paste an object without linking it.
- See the following two examples for Microsoft Word and Excel:

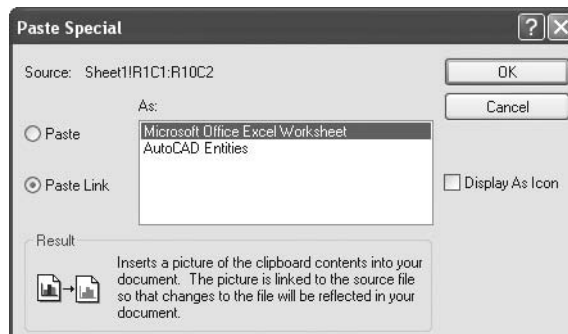
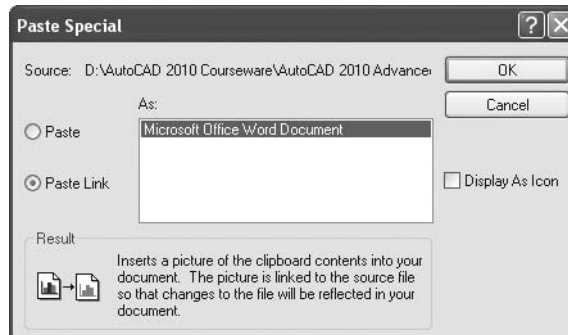


- You can select the type of pasting you require:
 - The source format (e.g., Word, Excel)
 - Different types of pictures (e.g., Metafile, Enhanced Metafile)
 - **AutoCAD Entities**
- Select the desired type and click **OK**.
- If you copied Microsoft Word objects, then you issue the **Paste Special** command in the AutoCAD drawing and select the **Paste** option and **AutoCAD Entities**. The objects will be pasted as **Single Line Text** objects.

- If you copied Microsoft Excel objects, then you issue the **Paste Special** command in the AutoCAD drawing and you select the **Paste** option and **AutoCAD Entities**. The object will be pasted as a table. If the Excel table has any formulas inside the cells, AutoCAD will preserve the same formulas to be used in the drawing.

Paste Link

- The **Paste Link** option allows you to paste a link.
- See the following two examples for Microsoft Word and Excel:



- Microsoft Word has only one option: to paste links as MS Word entities, which will make them exactly like **OLE**.
- However, MS Excel has two options; the first is to paste as Excel entities (**OLE**), and the second is to paste as **AutoCAD Entities** (i.e., as a table).
- If you choose to paste an Excel sheet as **AutoCAD Entities**, then when you select it and hover over it, two symbols will appear: *locked*, and *linked*.
- You can unlock any number of cells in the table. Thus, you will be able to change the values and those changes will be reflected in the original Excel sheet.

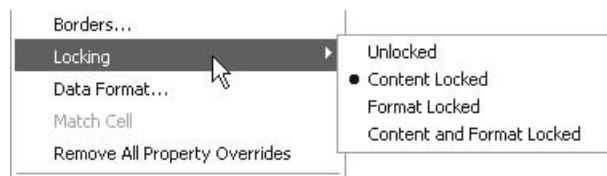
- Consequently, you can do two things:
 - Change the values in Excel and then update them in AutoCAD.
 - Change the values in AutoCAD and then update them in Excel.
- Perform the steps in the following example:
 - Copy a part of a sheet in Excel.
 - Using the **Paste Special** command with the **Paste Link** option, paste what you copied in AutoCAD as **AutoCAD Entities**.
 - Select the table (from the outside frame), then hover over it. You will see the two symbols we just mentioned. See the following example:

The screenshot shows an Excel table with two columns: 'Room #' and 'Area (sq. m.)'. The table contains data for rooms 201 through 209. A tooltip is displayed over the table, providing details about the data link.

Room #	Area (sq. m.)
201	11.70
202	11.70
203	11.70
204	
205	
206	
207	
208	
209	22.05

Data Link
 Excel Data Link 3
 C:\Documents and Settings\Munir\Desktop\Room Area.xls
 Link details: Range: A1:B10
 Last update: 04-Mar-09 5:09:25 PM
 Update status: Succeeded
 Update type: Updated from source
 Lock state: Content locked

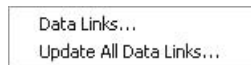
- For example, select the first two cells and then right-click. You will see a lengthy menu; select **Locking** and then **Unlocked**.



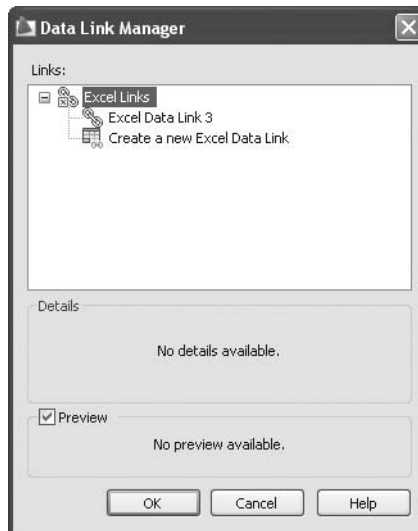
- Select one of the unlocked cells and change the value.
- At the lower right part of the screen, you will see a small icon called **Data Link**. See the following example:



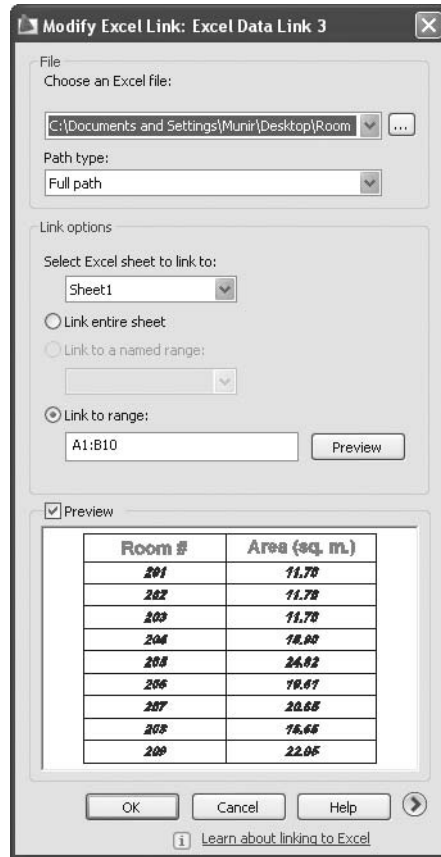
- Right-click and you will see the following menu:



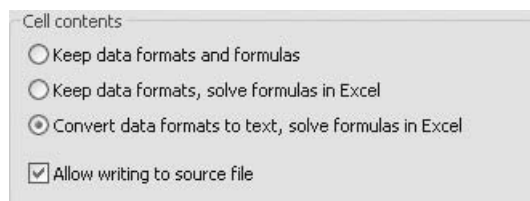
- Select the **Data Links** option and you will see the following dialog box:



- Double-click the **Excel Data Link** (the number varies depending on the number of links you are making in your current drawing). You will see the following dialog box:



- At the lower right side of the dialog box (beside the **Help** button), you will see a small button with an arrow pointing to the right. Click this button to get the advanced options of this dialog box. At the top right of this dialog box, you will see the following:



- Make sure that the option **Allow writing to source file** is on.

- Click **OK** and then **OK** again to close the two opened dialog boxes.
- Select the table again and right-click. Choose **Write Data Links to External Source**:



- The following prompt should appear:
- ```
1 object(s) found.
1 data link(s) written out successfully.
```
- If your Excel sheet is closed, then when you open it you will see that the value has changed.

## **IMPORTING OBJECTS FROM OTHER APPLICATIONS (GENERIC)**



### **Exercise 58**

- Start AutoCAD 2010.
- Open the file *Exercise\_58.dwg*.
- Make sure you are at the **ISO A1 Layout**.
- Start Microsoft Word and open the file *General Notes.doc*.
- Press [Ctrl] + A to select all text and then select **Edit/Copy**.
- Return to AutoCAD 2010. Make sure you are at layer **Text** and select **Edit/Paste**.
- Place the text in the upper right-hand corner of the layout.
- Change the font size to 12 = 6 and check the content of the object.
- Take a look at the object that appeared in your drawing. What is it? (Hint: select it and the **Quick Properties** will tell you.) \_\_\_\_\_
- Start Microsoft Excel and open the file *Room Area.xls*.
- Select the existing table and copy it.
- Return to AutoCAD 2010. Select the **Paste Special** command and the **Paste Link** option; choose **AutoCAD Entities**.
- Place the table below the text.
- Close the Excel sheet (*very important*).
- Select and hover over the table. You will notice the two symbols: **Locked** and **Linked**.
- Select the row containing Room #205 (the two cells), and then right-click and select **Locking/Unlock**.
- Change the value of the area from 24.82 to 24.92.

- Go to the right side of the **Status Bar** and right-click the **Data Link** icon. Select the **Data Links** option and when the dialog box appears, double-click **Excel Data Link 1**. Click the small arrow to the right and select the **Allow writing to source file** option to turn it on. Click **OK**, and then click **OK** again.
- Select the table again. Right-click and select **Write Data Links to External Source**.
- The following message should appear in the **Command Window**:

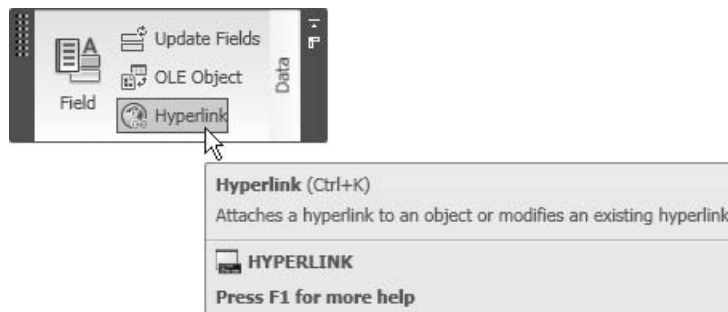
```
1 object(s) found.
```

```
1 data link(s) written out successfully.
```

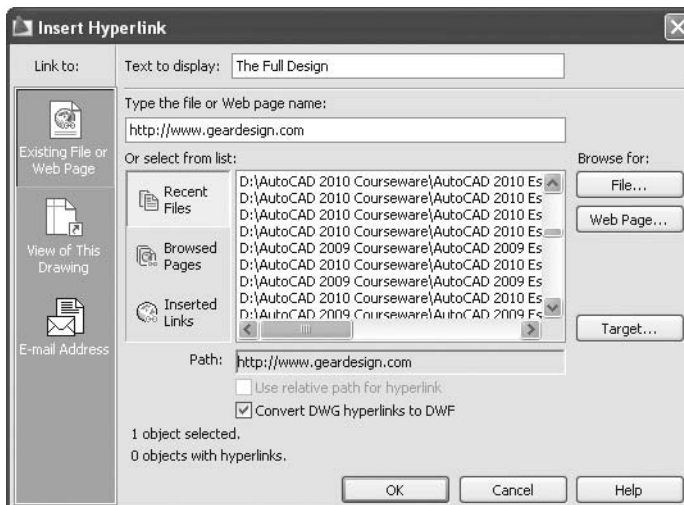
- If this is the case, then open the Excel sheet again and you will see that the value has changed.
- Save the file and close it.

## 12.5 THE HYPERLINK COMMAND

- AutoCAD drawing continues to become more sophisticated. It is no longer just composed of simple graphical objects—it is full of data and information.
- Using the **Hyperlink** command, you can link any object(s) with any type of data (e.g., websites, email addresses, other AutoCAD drawings, Word, Excel, and PowerPoint® files).
- The procedure for adding a hyperlink is very simple:
  - Select the desired object(s) in AutoCAD.
  - Make sure you are on the **Insert** tab on the **Ribbon** and, using the **Data** panel, select the **Hyperlink** button:



- The following dialog box will appear:



- In the **Text to display** window, type in the tooltip you wish to show the user once he or she gets closer to these object(s).
- In the **Type the file or Web page name** window, type the website address or the file path name.
- Click **OK** when you are done.



- You can do this in reverse order. Select the objects from the application you want and select **Edit/Copy**. Go to the desired AutoCAD drawing and select **Edit/Paste as Hyperlink** and then choose the objects in AutoCAD that you want linked. Now your AutoCAD objects are hyperlinked.

## THE HYPERLINK COMMAND (GENERIC)

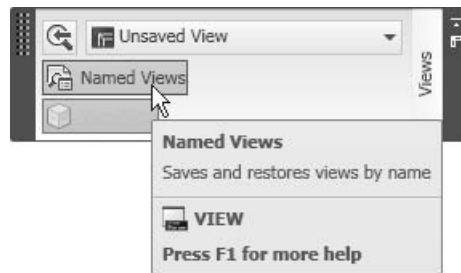


### Exercise 59

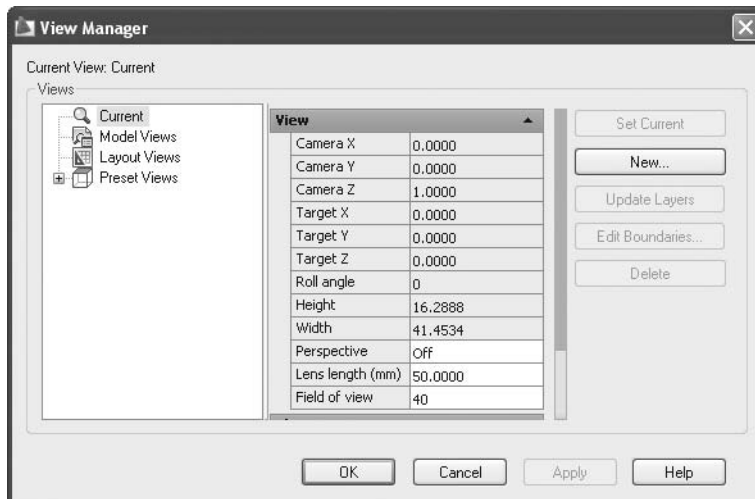
- Start AutoCAD 2010.
- Open the file *Exercise\_59.dwg*.
- Select the three-dimensional shape and press [Ctrl] + K. The **Hyperlink** dialog box will appear.
- Type PART DETAIL in the **Text to display** window.
- Click the **File** button and select the file *PART DETAIL.dwg*.
- Click **OK**.
- Test the hyperlink.
- Save the file and close it.

## 12.6 THE VIEW COMMAND

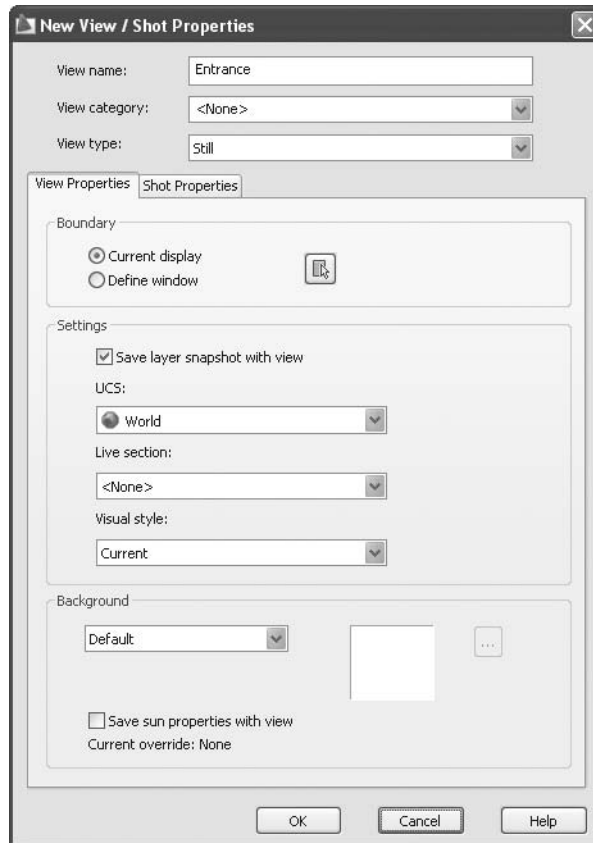
- A **View** in AutoCAD is any portion of the drawing saved under a name.
- There are two ways to define a view in AutoCAD:
  - Zoom to the required area using the **Zoom** and **Pan** commands, issue the **View** command, and create and name a new view.
  - Issue the **View** command to create a new view. In the dialog box specify two opposite corners, whatever is inside the window will be named as the view.
- To issue the **View** command, make sure you are on the **View** tab on the **Ribbon** and, using the **Views** panel, click the **Named Views** button:



- The following dialog box will appear:

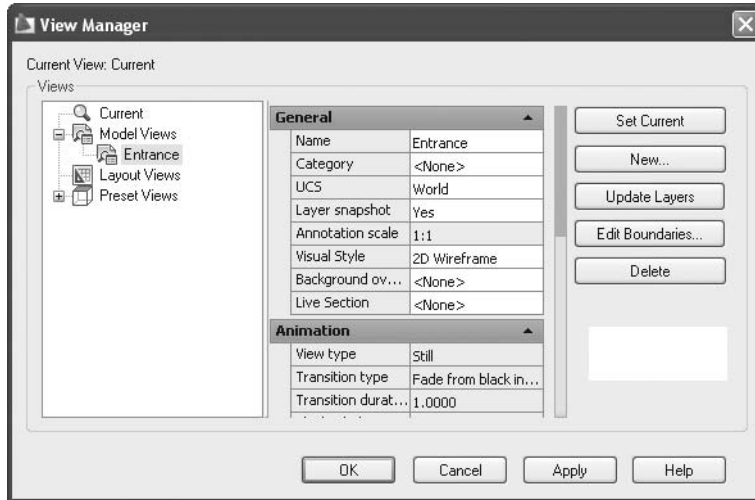


- Click the **New** button and the following dialog box will appear:



- Specify the **View name**.
- Specify the **View category**, which is user-defined. You can type the name of the category as you wish or you can leave it empty. Then, specify **View type** (AutoCAD 3D).
- From **Boundary** select either **Current display** (you have already zoomed to the desired area of the drawing) or **Define window** (you will now zoom to the desired area). Then, click on the small button at the right.
- Under **Settings**, choose **Save layer snapshot with view**, which means you will save the current status of the layers with the view so when you retrieve it you will restore this layer status with it. The other three options related to the 3D options in AutoCAD 2010 are: **UCS**, **Live Section**, and **Visual Style**.

- Under **Background**, select the background of the view. You can select the default color (the current background color), or you can select **Solid**, **Gradient**, **Image**, or **Sun & Sky** (these are also related to the 3D options in AutoCAD 2010).
- Click **OK** to save the view.
- After you save several views you will see the following dialog box:

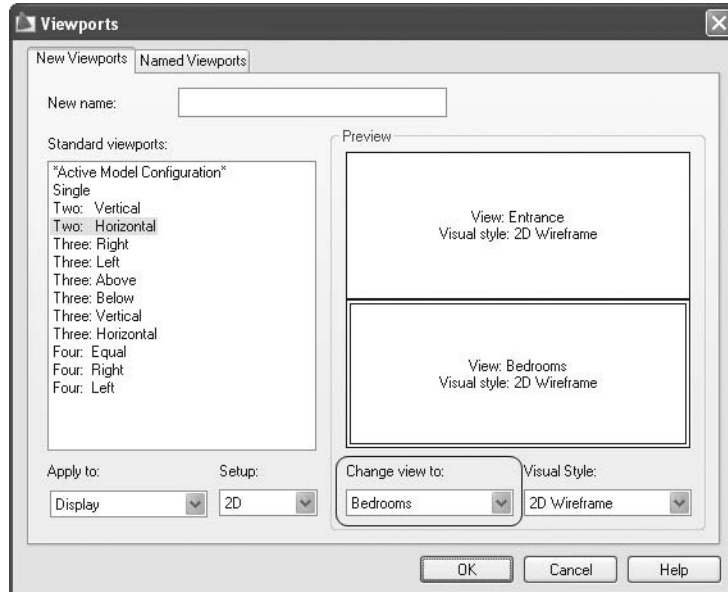


- Select the name of the view and click the **Set Current** button to restore the selected view.
- Click the **Update Layers** button if you want to restore the layer status saved with the view.
- Click the **Edit Boundaries** button if you want to change the window containing the view.
- To remove the view from your drawing, select the name of the view from the list and click the **Delete** button.



## Viewports

- You can utilize the views you created in the **Viewport** dialog box.
- Start the **Viewport** dialog box:



- As you can see in the **Preview** area, you can select each part of the screen.
- After making your selection, look at the lower part of the dialog box where it says **Change view to**, and select the saved view that you wish to show in the selected viewport.

## THE VIEW COMMAND (GENERIC)



### Exercise 60

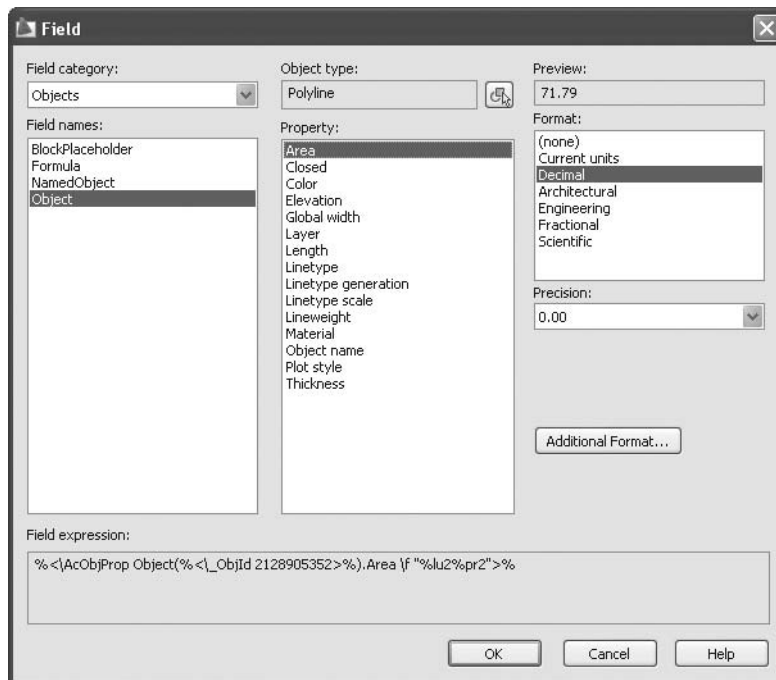
- Start AutoCAD 2010.
- Open the file *Exercise\_60.dwg*.
- Create two views for Room #208 and Room #209.
- Go to **ISO A1 Layout** and make layer **Viewports** current.
- Under **Standard viewports**, specify **Two: Horizontal**, showing the two views.
- Save the file and close it.



- By default the **Field category** is **All**, but you can select the desired field category from the following:
  - **Date & Time** includes everything related to the date and time of the drawing.
  - **Document** includes everything related to the AutoCAD file.
  - **Linked** includes everything related to hyperlinks.
  - **Objects** includes everything related to AutoCAD objects.
  - **Other** includes everything related to AutoCAD programming languages or system variables.
  - **Plot** includes everything related to the plotting process in AutoCAD.
  - **Sheet Sets** includes everything related to the Sheet Sets.

### Example 1

- The following is the **Field** dialog box:



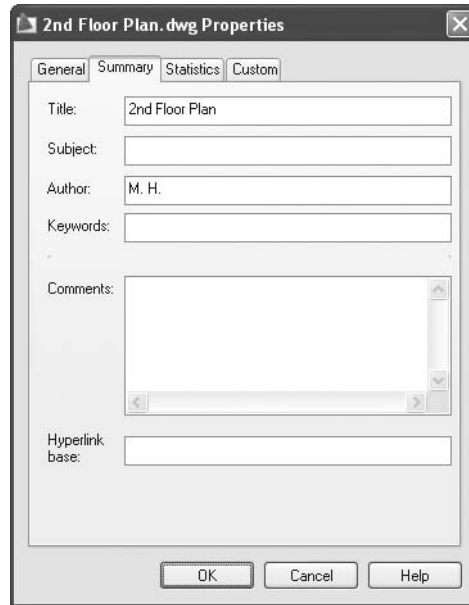
- The selections made in the example dialog box are: **Field category** is **Objects**, the **Field name** is **Object**, and **Polyline** is the **Object type**. AutoCAD lists the **Properties** of this specific object. From these properties we selected **Area**. In the **Preview** for the area we changed the **Format** of the area to be **Decimal**, and the **Precision** to be **0.00**.



- The field will be displayed using the current text style and current text height.

### Example 2

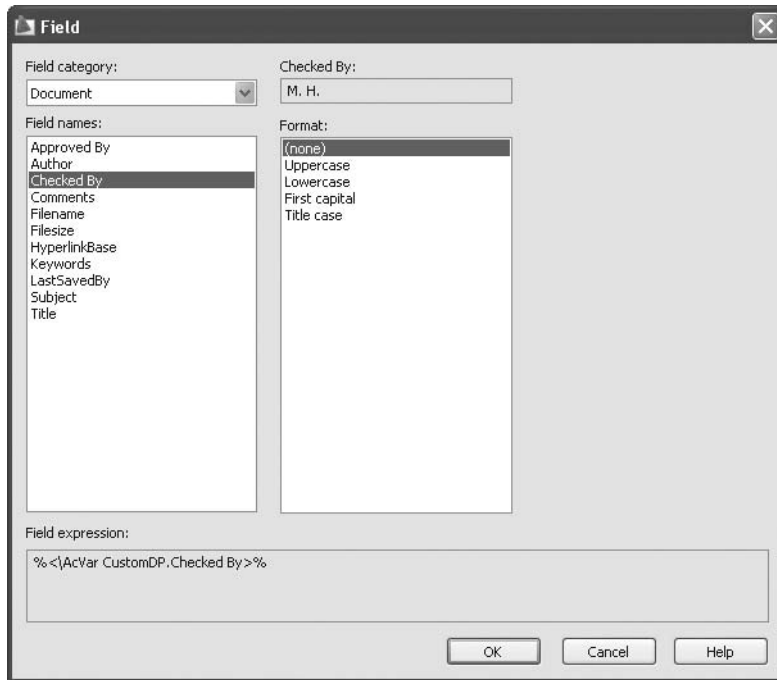
- From the **Application Menu**, select **Drawing/Drawing Properties**. You will see the following dialog box:



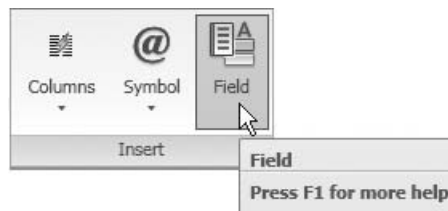
- Go to the **Summary** tab and fill in the required data like **Title**, **Subject**, and **Author**.



- Go to the **Field** command and select **Document** from the list. You will see something like the following:



- If you are inside the **MTEXT** editor, you can use the **Insert Field** button in the **Insert** panel, which appears with the **MTEXT** command.



- You can also right-click inside the **MTEXT** editor and select **Insert Field** from the menu. (Additionally, you can use [Ctrl] + F.)
- You can do the same thing using the **TEXT** command.
- In the **Attribute Definition** dialog box there is a button to insert a field (this command will be discussed in the coming chapters).
- When using the **Table** command, you can double-click inside a cell and right-click to select the **Insert Field** option.



- By default, a field similar to the following illustration will be inserted with a background:

M. H.

- If you don't wish to have a background, click the **Options** button at the bottom of the **Application Menu**, select the **User Preferences** tab, and click off the **Display background of fields** option.



## USING FIELDS (GENERIC)



### Exercise 61

- Start AutoCAD 2010.
- Open the file *Exercise\_61.dwg*.
- Check the following:
  - You are at the **Model Space**.
  - **Text** is the current layer.
  - **Standard** is the current **Text Style**.
- Using **Multiline Text** in the room at the upper left, type the following:
  - **Room #201** (in the first line)
  - **Area =** (in the second line)
- After the equals sign, right-click and select the **Insert Field** option.
- In the **Field category** select **Objects**. From the **Field names** select **Object**, and then select the *cyan polyline* around the room. Select **Area** and change the **Precision** to be 0.00, and then click **OK**. Finally, select the **Close Text Editor** button.
- Do the same for any other room in the drawing.
- From the **Application Menu**, select **Drawing/Drawing Properties** and change the following:
  - **Title** = 2nd Floor Plan
  - **Author** = M. H.
  - Go to **Custom**, define a new field **Checked By**, and enter its value as **A. A.**

- Go to **ISO A1 Layout**, and using the **Field** command, do the following:
  - Using text style **TitleBlock\_Regular** insert the title of the drawing in the right place.
  - Using text style **ISO Proportional**, insert the fields **Designed By (Author)** and **Checked By**.
- Remove the background of the fields.
- You should have something like the following:

| Itemref              | Quantity            | Title/Name, designation, material |              |
|----------------------|---------------------|-----------------------------------|--------------|
| Designed by<br>M. H. | Checked by<br>A. A. | Approved by - d                   | APPROVED_BY_ |
| 2nd Floor Plan       |                     |                                   |              |
| 14                   |                     |                                   |              |

- Save the files and close it.

## 12.8 FORMULAS AND TABLES

- Earlier in this book we covered how to create a table style, and how to insert a table in your drawing.
- Here we will discuss how to create formulas, imitating Microsoft Excel.
- Also, we will cover some advanced functions in **Tables**.

### Formulas

- For our purposes, we will assume you do not possess any prior MS Excel experience. Therefore, we will lay out some basic rules:
  - Each cell has an address linking it to its column and row. For example, A3 means this cell is in column A and in row 3.
  - To start a formula in any cell, double-click inside the cell, and always start your input with an equals sign (=). Input a cell address, along with a basic math function like +, -, \*, or /. For example: a valid formula is =A3+B3-C3 (this formula will be evaluated from left to right, unless there are brackets, which will be evaluated first).
  - Also, you can use predefined functions like: **SUM(range of cells)** and **AVERAGE(range of cells)**.



- The cell address is always relative in order to make our work easier. Assume we have the following formula in cell E14:  $= (A14 + B14) - (C14 / D14)$ . If you copy the contents of the cell to E15, what will happen? The formula will be  $= (A15 + B15) - (C15 / D15)$ . Why? Because the formula is reflecting relative addresses.
- You can add a dollar sign (\$), if you want to make the address absolute. For example, \$A3: column A will always be the column to copy from, but the row will change with the copying. A\$3 means row 3 will always be the row to copy from and the column will change. \$A\$3 means you will always go to cell A3.
- If you select the cell (by clicking inside its border) there will be a handle (**Autofill grip**) at the lower right corner that will allow us to copy the contents of the cell to the other cells. See the following example:

|   | A      |
|---|--------|
| 1 | Result |
| 2 | 10     |
| 3 | 20     |

Autofill grip

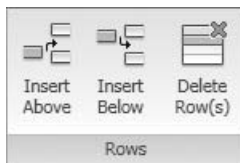
- Click on the handle and drag it downward (or upward), and the formula will be copied accordingly.
- You can add an **Autofill grip** to increment the numbers by one.
- Also, if you typed two numbers, select both cells and drag the **Autofill grip**. AutoCAD will take the difference between the two numbers and consider it as an increment, so if you drag the handle of the two cells an incremental number will always be shown.

## 12.9 ADVANCED TABLE FEATURES

- If you double-click inside a cell, you will be able to input data, or edit existing data. If you click on one cell, you will see four grips and the handle (**Autofill grip**). You will also see a new tab called **Table** and panels that contain some advanced functions. These are discussed in the following sections.

### Rows

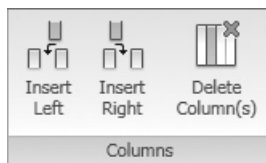
- You will see the following panel:



- **Insert Above** means a new row will be added above the selected row.
- **Insert Below** means a new row will be added below the selected row.
- **Delete Row(s)** will delete the selected rows.

### Columns

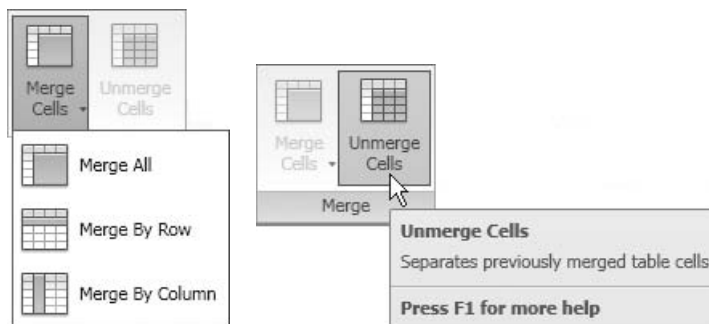
- You will see the following panel:



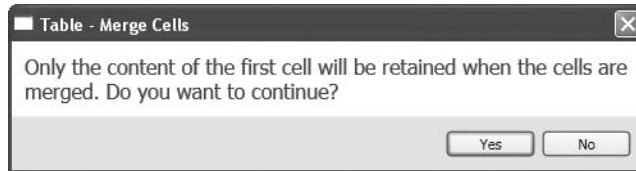
- **Insert Left** means a new column will be added to the left of the selected column.
- **Insert Right** means a new column will be added to the right of the selected column.
- **Delete Column(s)** will delete the selected columns.

### Merge

- You will see the following panel:



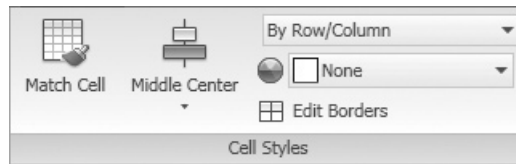
- For this command to be active, you have to select more than one cell. This function is used to **Merge** cells. If you select several cells, AutoCAD will give you three options: **All** (to merge all cells in single cell), **By Row** (to merge all cells in the same row together while keeping the columns), or **By Column** (to merge all cells in the same column together while keeping the rows). In all cases the following dialog box will appear before the merging process takes place:



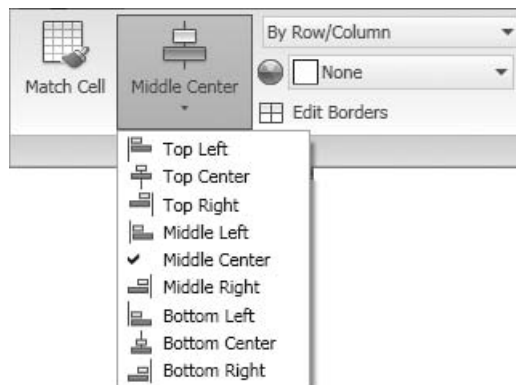
- Unmerged Cells** is the opposite of the **Merge** function. You can unmerge merged cells whenever you want, even after closing the file.

### Cell Styles

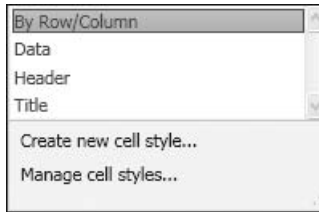
- You will see the following panel:



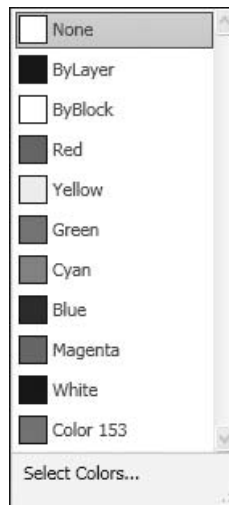
- When using the **Match cell** function, the user will first select the cell that AutoCAD will copy the properties from, and then the user will select as many cells as he or she wishes it to match with.
- Select the **Alignment** button. You will see the following list:



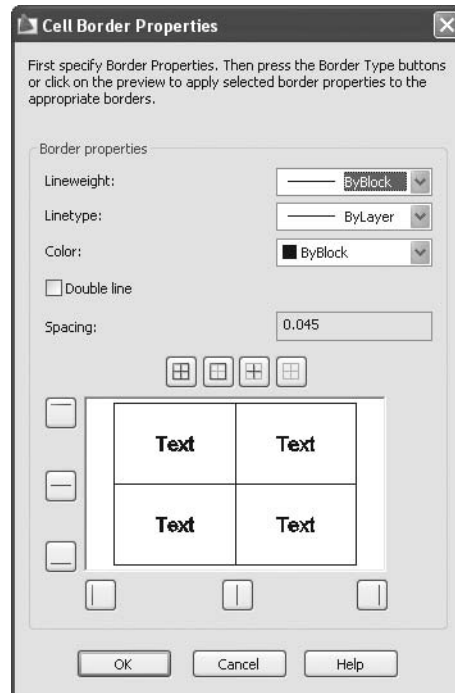
- This is the **Alignment** of the text related to the borders of the cell. There are nine choices: **Top Left**, **Top Center**, **Top Right**, **Middle Left**, **Middle Center**, **Middle Right**, **Bottom Left**, **Bottom Center**, and **Bottom Right** (the shape of the button will change depending on the alignment choice you make).
- Select the desired cell style for the selected cells. You will see the following list:



- Select the desired background color for the selected cells. You will see the following list:



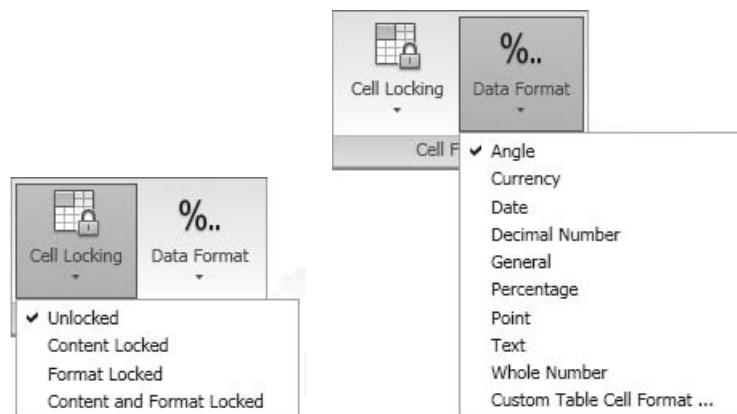
- Select **Edit Borders**. You will see the following dialog box:



- You must specify **Lineweight**, **Linetype**, and the **Color** of the borders. By default, the border is a single line, but you can change it to a double line. Specify the **Spacing** of the double line. Determine whether to make the lines for **All Borders**, **Outside Borders**, **Inside Borders**, or **No Borders**.

### Cell Format

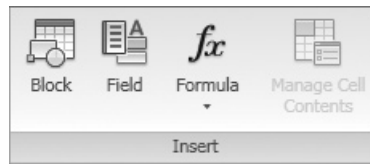
- You will see the following panel:



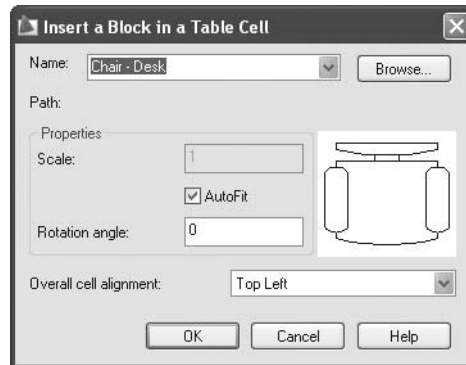
- This is the **Locking** function. You can elect to lock the cell so it cannot be changed. There are four different methods of locking: to lock the **Content** of the cell, to lock the **Format** of the cell, to lock the **Content and Format** of the cell, or to leave the cell **Unlocked**.
- To control the **Data Format**, you have nine choices to select from.

### Insert

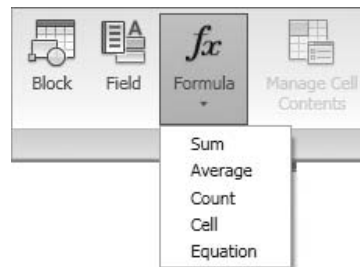
- You will see the following panel:



- When you click on the **Block** button, you will see the following dialog box:



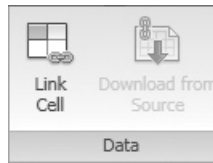
- Specify the **Name** of the block, **Scale** or **AutoFit**, the **Rotation** angle, and the **Alignment** of the block reference to the desired cell.
- The **Field** button is used to insert a field in the cell (this has been discussed previously).
- When using the **Formula** button, you will see the following list:



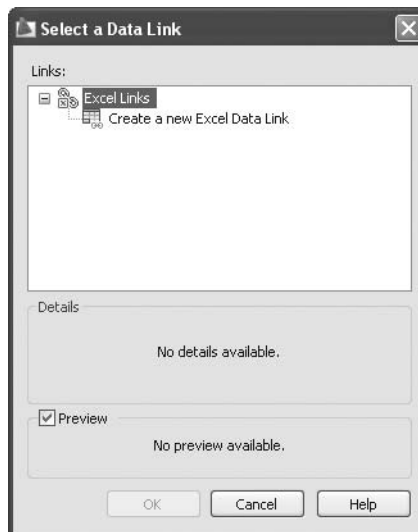
- You will insert one of the predefined formulas from a list of five choices.

**Data**

- You will see the following panel:



- If you click the **Link Cell** button, you will see the following dialog box:



- This box is the same dialog box we discussed when we introduced the concept of importing Microsoft Excel objects to an AutoCAD drawing.

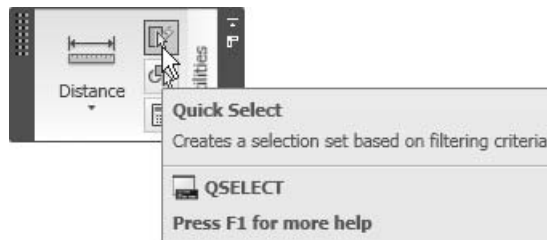
**FORMULAS AND ADVANCED FEATURES IN TABLES (GENERIC)****Exercise 62**

- Start AutoCAD 2010.
- Open the file *Exercise\_62.dwg*.
- Check the preexisting table.
- Select all the **Data cells** and make the **Alignment** = Middle Center.
- Select the rightmost column.
- Insert a new column to its right.
- Make the **Header of the column** = Area.

- In the first cell, below the header, type the following formula: **=C3\*D3**.
- Copy the formula to the cells below.
- Select the lower row.
- Insert a new row beneath it.
- Select all cells of the new row except the rightmost cell.
- Merge the cells by row.
- In the merged cells type **Total Area**.
- Click the rightmost cell, and then click the **Formula** button and select the **Sum** function.
- Select the range of the areas (from E3 to E6).
- Save the file and close it.

## 12.10 THE QUICK SELECT COMMAND

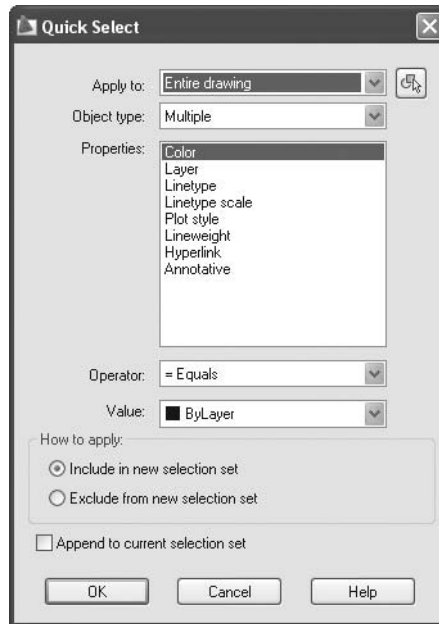
- The selecting methods covered previously were all visual methods, which you must either click on an object or specify some sort of window to select a group of objects.
- The **Quick Select** method will allow you to select objects based on their properties, even if these objects are not easily seen throughout the drawing.
- **Quick Select** works closely with **Properties** as it supplies the objects.
- To issue the **Quick Select** command, make sure you are on the **Home** tab on the **Ribbon** and, using the **Utilities** panel, click the **Quick Select** button:



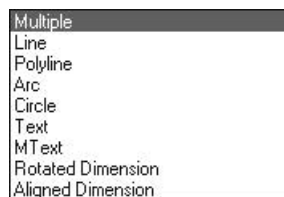
- Another way to issue the command is to right-click while you are in the **Graphical Area** and choose **Quick Select** from the displayed short-cut menu.



- Either method will produce the following dialog box:



- In the **Apply to** section, select whether to search the **Entire drawing** or a specific window.
- By default, AutoCAD knows all of the types of objects in any file. Hence, in the **Object type**, you may select **Multiple** or any specific object type that exists in your drawing. See the following illustration:

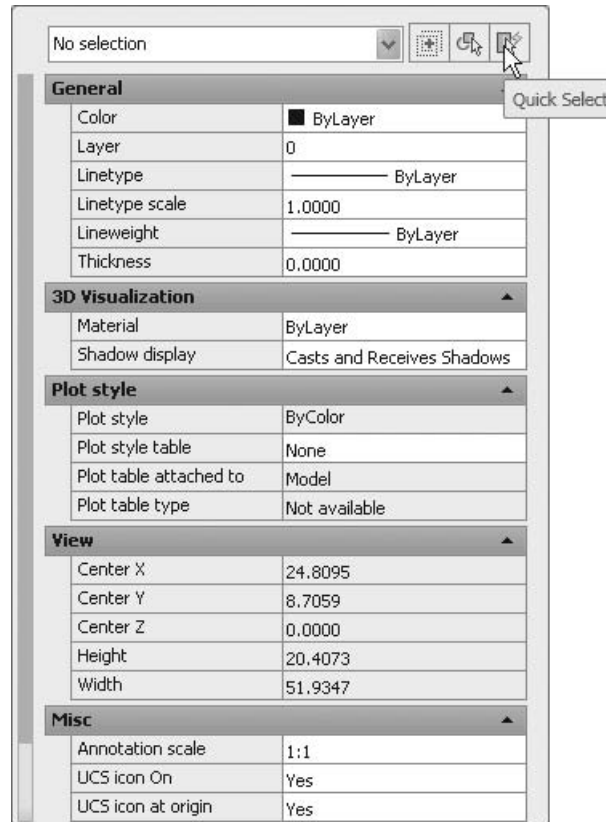


- **Properties** differ according to the object type:
  - If you chose **Multiple**, you will only see a list of general properties.
  - For example, if you had a polyline in your drawing, then you will see both the general properties and the specific properties of polylines.
- Specify the **Operator** (**Equals**, **Not Equal**, **Greater than**, **Less than**, or **Select All**), and its **Value**.
- Under **How to apply**, you have two choices:
  - Create a new selection set based on the objects that fit the criteria mentioned in the **Quick Select** dialog box.
  - Select all the objects that don't fit the criteria.

- You can also elect to **append** the selected objects to an existing selection set (the selection set must be created before issuing the **Quick Select** command).



- You can also issue the **Quick Select** command from the **Properties** palette. See the following illustration:



## THE QUICK SELECT COMMAND (GENERIC)



### Exercise 63

- Start AutoCAD 2010.
- Open the file *Exercise\_63.dwg*.
- The people who drafted this drawing made the following mistakes:
  - All circles should have a **Diameter** = 0.24, but only one of 14 has the correct diameter; the others are all incorrect.
  - The dimensions **Rotated** (which is **Linear**), **Diameter**, and **Radial** are all in layer 0 (zero).

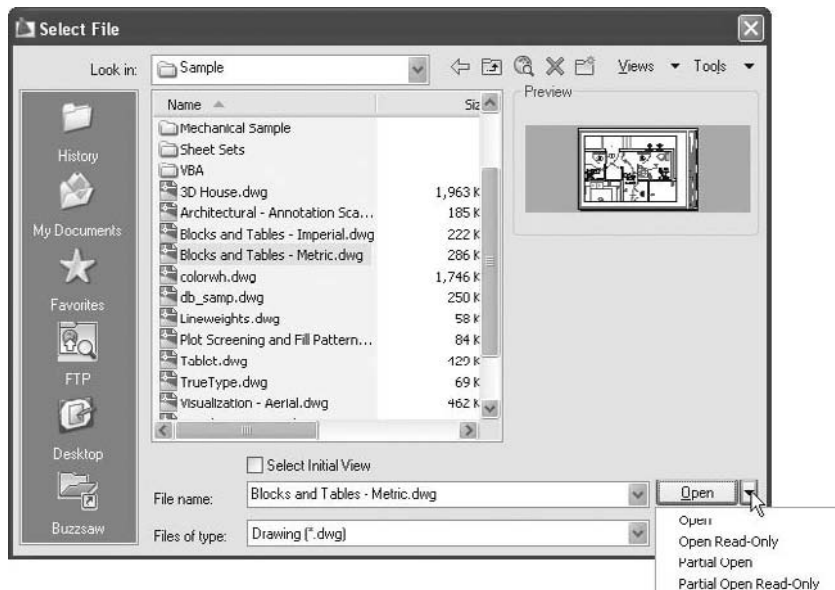
- With the **Quick Select** command and the **Properties** command, correct these two mistakes using the following guidelines:
  - Select all circles with **Diameter** < 0.24 and set the **Diameter** = 0.24.
  - Start the **Properties** command and click the **Quick Select** button.
  - Since there are three types of dimensions, select the first type, and use **Append to current selection set**.
  - Each time you select a dimension, make sure the value of the **Operator** is set to **Select All**.
  - Move all dimension objects to layer = **Dimensions**.
  - All center marks are in layer 0 (zero). Select all objects (**Multiple**) in layer 0 and send them to layer = **Dimensions**.
- Save the file and close it.

## 12.11 THE PARTIAL OPEN AND PARTIAL LOAD FEATURES

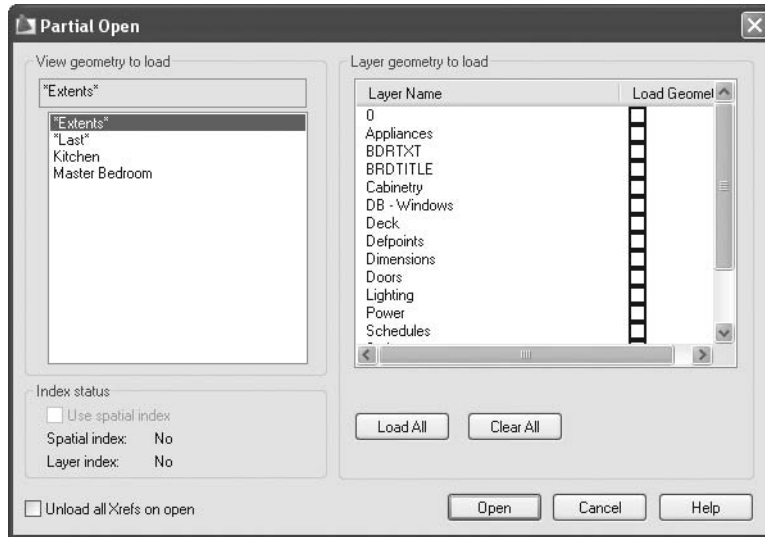
- Sometimes you have a massive drawing containing many views and layers, and you don't want to open it all, just part of it. In this case, you can use the **Partial Open** feature.
- Once you have opened a file as partial, you can partially load additional views and layers.

### Procedure

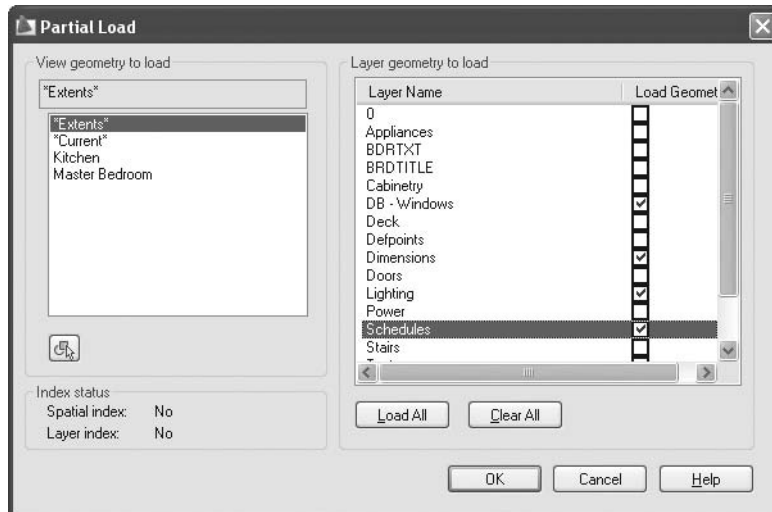
- Using the **Open** command, select the file name, and then click the small arrow at the right and select **Partial Open**. See the following dialog box:



- Once you select **Partial Open**, the following dialog box will appear:



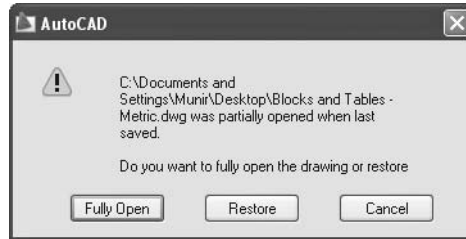
- You can select layers to be loaded for the “**Extents**” of the drawing.
- You can select specified layers to be loaded only in one of the saved views.
- Once you have partially loaded a file, you can type **partialload** in the **Command Window**. The following dialog box will appear:



- This dialog box is identical to the **Partial Open** dialog box, except for the small button at the left. By default, you can partially load the layers in the entire drawing or you can specify a window in the drawing to load the extents of objects.



- If you save your file and close it while it was partially opened (or loaded), once you try to open it again AutoCAD will display the following message:



- Select whether you want it fully opened or want to restore the status as it was the last time you saved it.

## THE PARTIAL OPEN AND PARTIAL LOAD FEATURES (GENERIC)



### Exercise 64

- Start AutoCAD 2010.
- Using the **Open** dialog box, select *Exercise\_64.dwg* (don't double-click it).
- Using the **Open** button, select **Partial Open**.
- Select the following layers:
  - **Wall**
  - **Door**
  - **Partition**
- Click **OK** to open the file with these layers only.
- Start the **Partial Load** command.
- Select the view named **Room #209** and the layer **Text**.
- Save the file and close it.
- Try to open it again. What is AutoCAD's response?

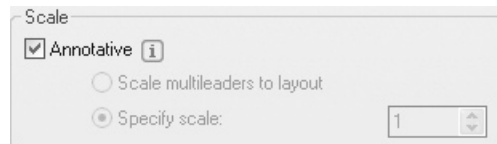
## 12.12 THE ANNOTATIVE FEATURE

- We always recommend that AutoCAD users print from layouts and not from **Model Space**. (Although you can print from **Model Space**!)
- It is obvious that Autodesk® will not develop anything new for people who like to print from **Model Space**, and since AutoCAD 2000, Autodesk has been putting all of its resources into printing from layouts.
- If we assume that we will print only from layouts, then we will use viewports (there is no other way), and viewports need VP scale (as we discussed earlier).

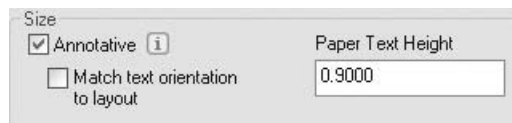
- If the preceding is true, then we have a problem. The problem lies in your annotation objects, such as:
  - **Text** (**Single line** and **Multiple lines**)
  - **Dimensions** (**Dimensions** and **Multileader**)
  - **Hatching**
- The normal procedure is as follows:
  - Specify all of your annotation in the **Model Space**.
  - Go to **Layout**, **Create Viewports**, and scale them.
  - Since you will have different viewports in the same layout with different scales, annotation objects will appear clear and readable in some viewports and tiny and unreadable in others.
  - Users tend to control the visibility of the layers in the viewports by creating multiple styles with different sizes and by inputting different sizes in different layers.
- In order to solve this problem, Autodesk invented a feature for the annotation object, called **Annotative**.
- You will find the **Annotative** feature in different places:
  - In the **Fit** tab of the **Dimension style** box, you will see the following:



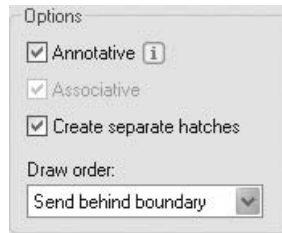
- In the **Leader Structure** tab of the **Multileader** style box, you will see the following:



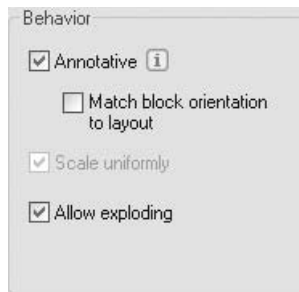
- In the **Text style** box, you will see the following:



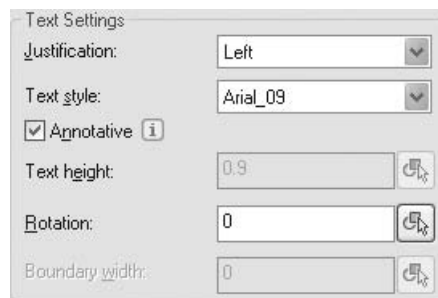
- In the **Hatch** dialog box, you will see the following:



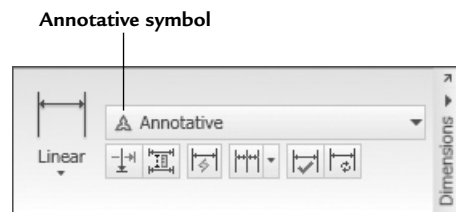
- When using the **Block** command, you will see:



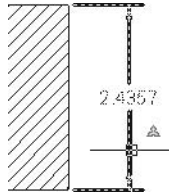
- When using the **Attribute** command, you will see the following:



- To differentiate an annotative style from an ordinary style, check the symbol at the left in any pop-up list, just like the following:

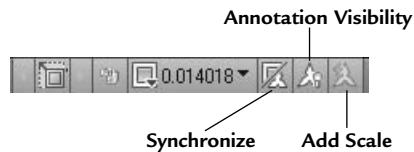


- To know whether an object is annotative or not, hover over it and you will see the following:



### Procedure

- In order to utilize the **Annotative** feature in the best way, follow this procedure:
  - Create your drawing in **Model Space**, but without **Dimensions**, **Text**, **Hatching**, and so on.
  - Go to your layout and add the desired viewports.
  - Select the desired viewport to scale (either by selecting the border or by double-clicking inside it).
  - Set the scale of the viewport using the **Status Bar**:



- Click the scale pop-up list and choose the desired scale. This scale will be used for both the annotation objects and the viewport.
- Double-click inside the viewport to make it active.
- Use your annotative style to annotate your drawing.

### Important Notes

- By default, the following are true about annotation using annotative styles:
  - Once you input any annotation in the viewport, the annotation will appear only in this viewport and will not appear in other viewports unless they have the same scale value.
  - If you change the scale of the viewport, the annotation will disappear.
  - If you want to show the annotation in more than one viewport having different scale values, right click on the **Annotation Visibility** button and the following menu will appear:

- Show Annotation Objects for Current Scale Only
- Show Annotation Objects for All Scales



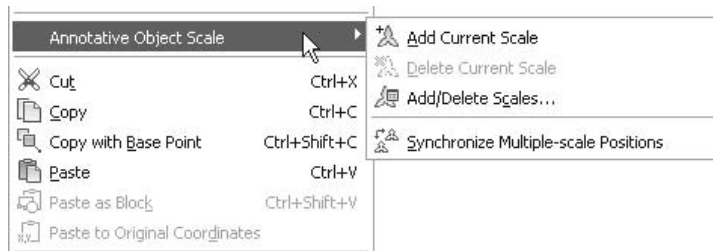
- Select between **Show Annotation Objects for Current Scale Only** or **Show Annotation Objects for All Scales**.
- You can also control how to add scale values to the viewport and whether they are **Automatic** or **Manual**. Right-click on the **Add Scale** button and the following menu will appear:

Automatically add scales to annotative objects when the annotation scale changes

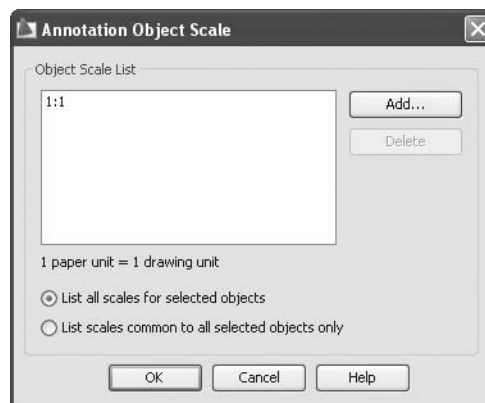
- If you used the **Zoom** command inside the viewport, then this action will change the scale value of the viewport. To correct this issue, click the **Synchronize** button and the following message will appear:

Annotation scale is not equal to viewport scale: Click to synchronize

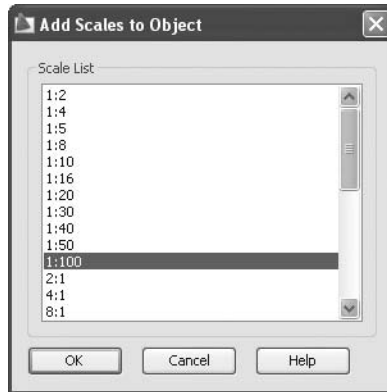
- When you click this message, your viewport scale will be restored.
- At any time you can use the lock in the **Status Bar** to lock the viewport scale.
- If you want to make an annotation object appear in different viewports that have different scales, then select it and right-click. When the menu appears, select **Annotative Object Scale**. The following menu will appear:



- Select the option **Add/Delete Scales**. The following dialog box will appear:



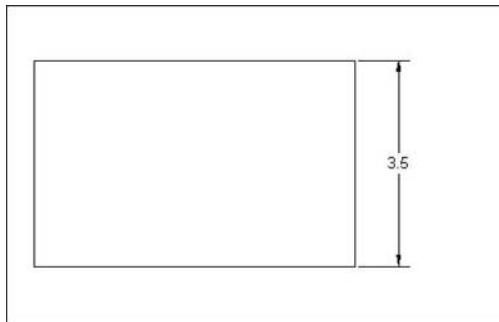
- Click the **Add** button. The following dialog box will appear:



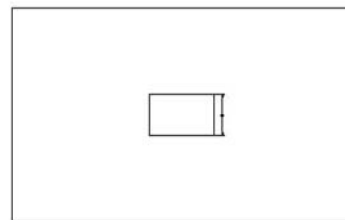
- Select the desired scale value and click **OK**, and **OK** again, to finish the command.

#### Before and After

- In order to appreciate this feature in AutoCAD, we have to see the before and after methods:
  - When you put your dimension objects in **Model Space** and you want to print from layout, you will get something like this picture (before):

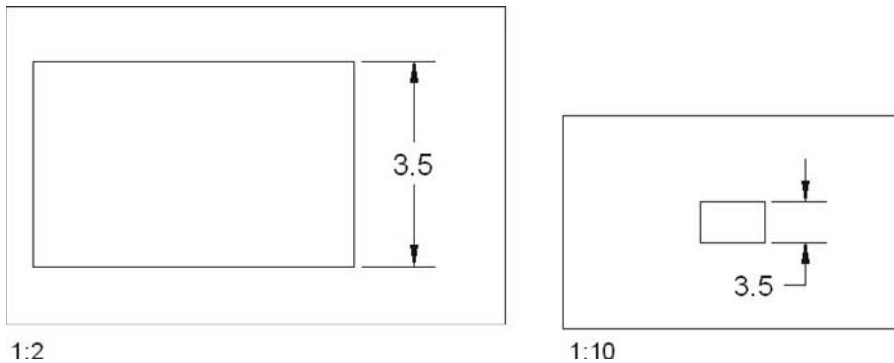


1:2



1:10

- As you can see, it is very clear in the 1:2 scale, but you can't see it in the 1:10 scale.
- When you make your dimension style **Annotative**, and you put your dimensions in layout, you will get something like this picture (after):

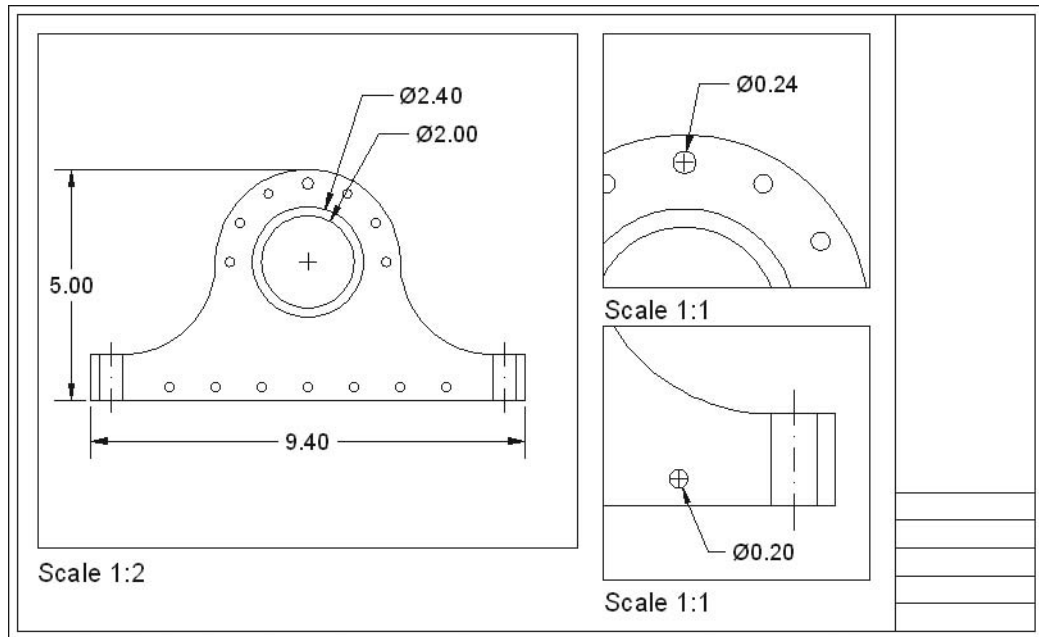


## THE ANNOTATIVE FEATURE (GENERIC)

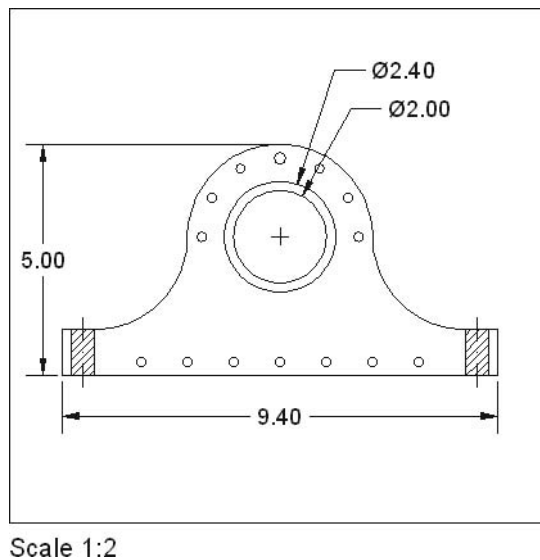


### Exercise 65

- Start AutoCAD 2010.
- Open the file *Exercise\_65.dwg*.
- Go to **A4 Landscape** layout.
- You will find three viewports—each one with its scale shown in the following illustration.
- Make the current layer **Dimensions**.
- Make the current dimension style **Annotative**.
- Make sure that you double-click inside each viewport before you input the dimensions. Now, input the dimensions as shown in the following illustration:



- Make the current layer **Hatch**.
- Double-click inside the viewport at the left and start the **Hatch** command. Be sure to switch **Annotative** on. Hatch the two holes around the center line. See the following illustration:



- Notice that the hatch didn't appear in the lower viewport!
- Select the hatch, right-click, and select **Annotative Object Scale/Add Delete Scales**. Add scale 1:1. What happens? \_\_\_\_\_
- Save the file and close it.

## CHAPTER REVIEW

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1. Which of the following statements is false?
  - a. You can offset objects and fillet them at the same time.
  - b. You can access **Quick Select** from the **Properties** palette.
  - c. You can open more than one file in AutoCAD, but you can't use the drag-and-drop technique to copy objects from one file to another.
  - d. **Partial Load** can be used only if the file was opened in **Partial Open**.
2. In order to select all objects of a certain type, at \_\_\_\_\_, use the **Select All** option.
3. Which of the following statements about **Tables** in AutoCAD is false?
  - a. You can insert a new column to the right of an existing column.
  - b. You can add a formula with relative or absolute references.
  - c. You can insert a field inside one of the cells.
  - d. You can't lock a cell in a table in order to prevent editing.
4. The total length of a polyline can be a field that you can insert in your drawing.
  - a. True
  - b. False
5. Using the **Edge** option in the **Trim** command, you can trim objects based on an extrapolated cutting edge.
  - a. True
  - b. False
6. Using \_\_\_\_\_, you can link any object in any drawing to another file.

**CHAPTER REVIEW ANSWERS**

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1. c
2. **Operator**
3. d
4. a
5. a
6. **Hyperlink**

# Chapter 13

# PARAMETRIC CONSTRAINTS AND ADVANCED BLOCK TECHNIQUES

## In This Chapter

- ◇ Parametric Constraints: An Introduction
- ◇ Geometric Constraints
- ◇ Dimensional Constraints
- ◇ Dynamic Blocks: An Introduction
- ◇ How Do You Recognize a Dynamic Block?
- ◇ Getting Started with Dynamic Blocks
- ◇ The Authoring Tool Palette for Dynamic Blocks
- ◇ Block Attributes: An Introduction
- ◇ Defining Block Attributes
- ◇ Inserting Blocks with Attributes
- ◇ Block Attributes: Controlling the Visibility
- ◇ Block Attributes: Editing Values (One-by-One)
- ◇ Block Attributes: Editing Values (Global)
- ◇ Redefining Attributes
- ◇ The **Synchronize** Command
- ◇ Extracting Attributes

## **13.1 PARAMETRIC CONSTRAINTS: AN INTRODUCTION**

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- Using the **Parametric Constraints** feature in AutoCAD® allows you to do two things:
  - Assign a geometric relationship between two objects that the user cannot change.
  - Assign a size to an object that the user cannot modify.
- This feature used to be available in the vertical products only, but now it is available in AutoCAD as well.
- This feature will help any engineer draw his or her design exactly as it was intended.

- With the power of parametric constraints, the designer will prevent any mistakes his or her team may make in later stages of the design process.
- There are two types of parametric constraints:
  - **Geometric**
  - **Dimensional**

## 13.2 GEOMETRIC CONSTRAINTS

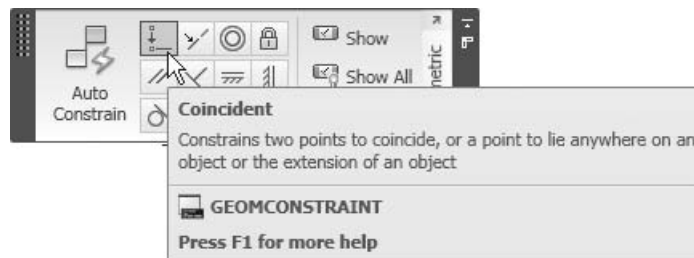
- In order to utilize these constraints, select the **Parametric** tab and use the **Geometric** panel:



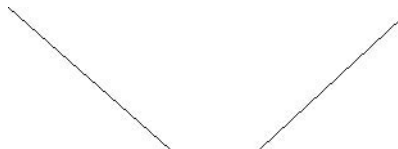
- As you can see, there are 12 geometric constraints, which are **Coincident**, **Colinear**, **Concentric**, **Parallel**, **Perpendicular**, **Horizontal**, **Vertical**, **Tangent**, **Smooth**, **Symmetric**, **Equal**, and **Auto Constrain**.

### Coincident

- Using the **Geometric** panel, click the **Coincident** button:



- The **Coincident** button will allow you to constrain two points to coincide.
- For example:
  - Assume we have the following two lines:

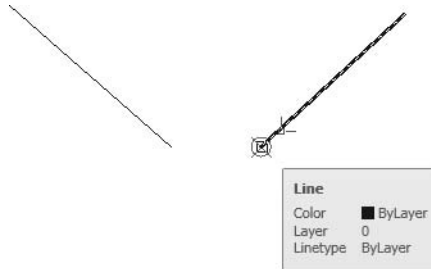




- Start the **Coincident** command. The following prompt will appear:

Select first point or [Object/Autoconstrain] <Object>:

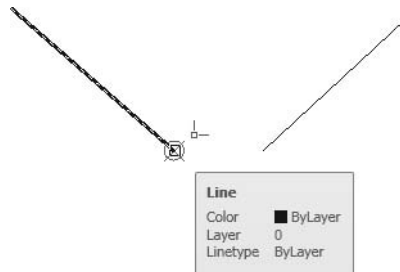
- Click a point on the first object as shown:



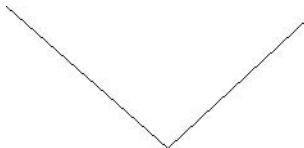
- The following prompt will appear:

Select second point or [Object] <Object>:

- Click the coincident point in the second object as shown:

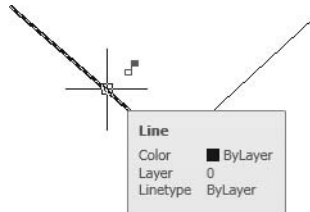


- The result will be as follows:



- Now if you try to move one of the lines and not the other one, the one to be moved will be stretched.

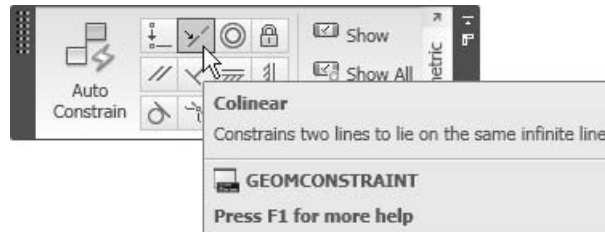
- If you hover over one of the two lines, you will see the following:



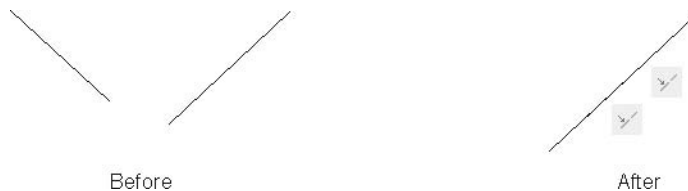
- This means that these two objects have parametric constraints placed on them.

### Colinear

- Using the **Geometric** panel, click the **Colinear** button:

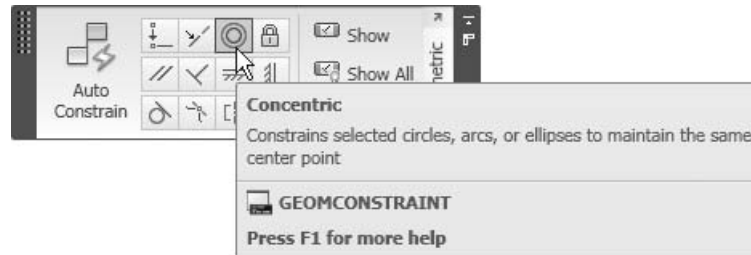


- The **Colinear** feature allows you to constrain two lines to lie on the same infinite line.
- See the following illustration:

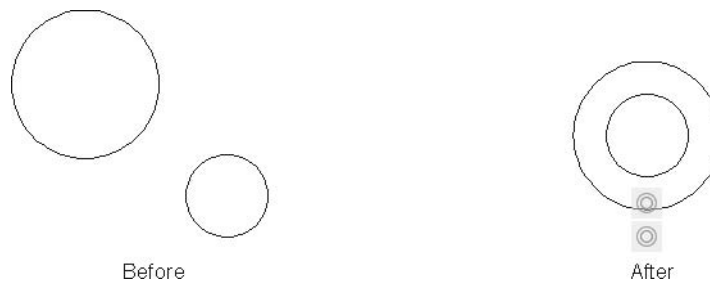


### Concentric

- Using the **Geometric** panel, click the **Concentric** button:

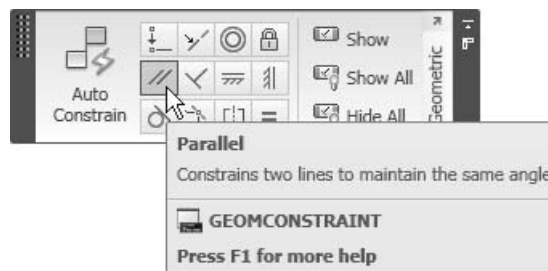


- The **Concentric** feature allows you to constrain selected circles, arcs, or ellipses to maintain the same center point.
- See the following illustration:



### Parallel

- Using the **Geometric** panel, click the **Parallel** button:

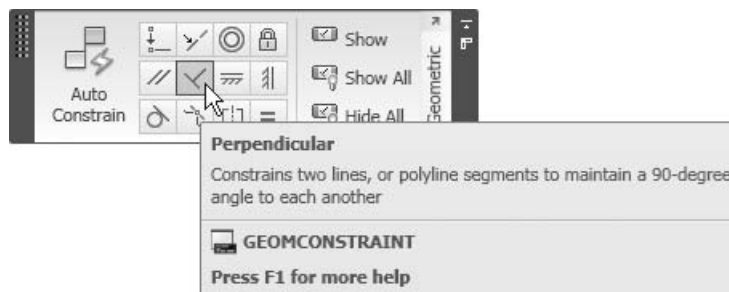


- The **Parallel** feature allows you to constrain two lines to maintain the same angle.
- See the following illustration:

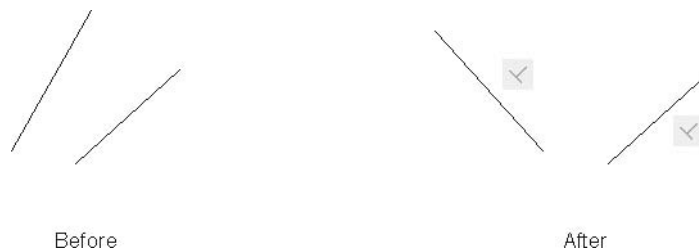


### Perpendicular

- Using the **Geometric** panel, click the **Perpendicular** button:

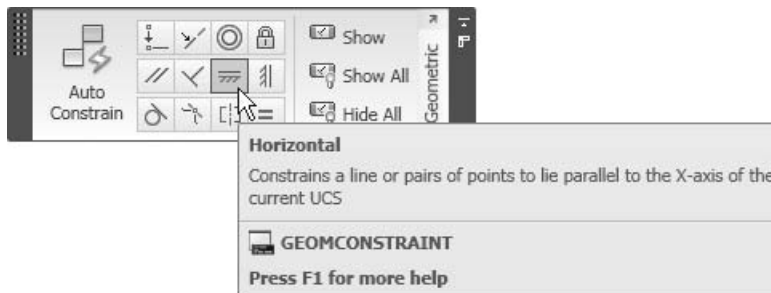


- The **Perpendicular** feature allows you to constrain two lines or polyline segments to maintain a 90° angle to each other.
- See the following illustration:



### Horizontal

- Using the **Geometric** panel, click the **Horizontal** button:

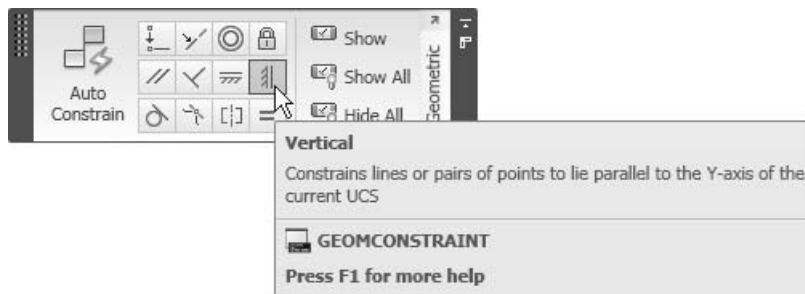


- The **Horizontal** feature allows you to constrain a line to lie parallel to the  $x$ -axis.
- See the following illustration:



### Vertical

- Using the **Geometric** panel, click the **Vertical** button:

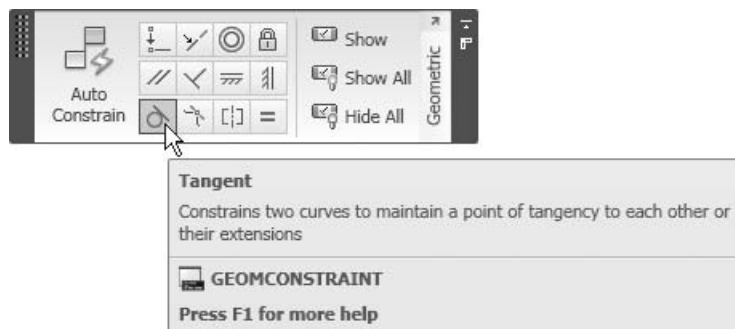


- The **Vertical** feature allows you to constrain a line to lie parallel to the  $y$ -axis.
- See the following illustration:

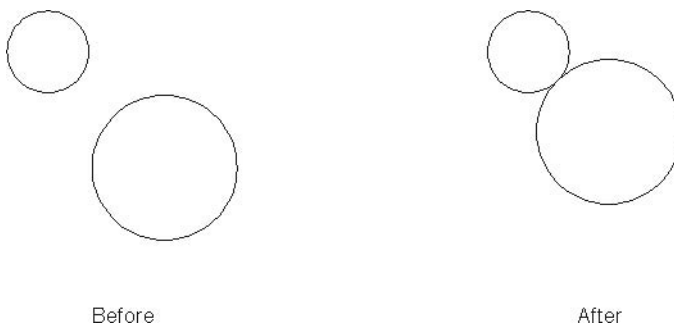


### Tangent

- Using the **Geometric** panel, click the **Tangent** button:

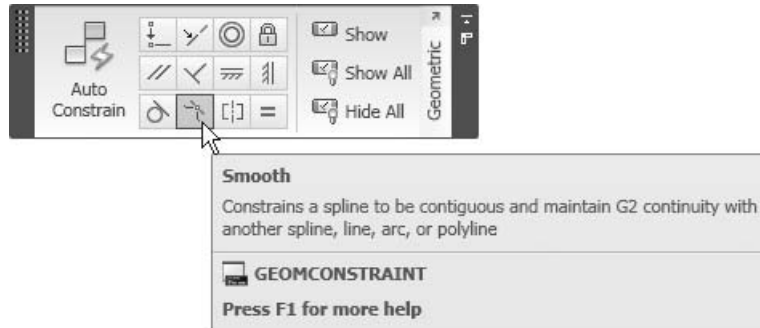


- The **Tangent** feature allows you to constrain two curves to maintain a point of tangency to each other.
- See the following illustration:



### Smooth

- Using the **Geometric** panel, click the **Smooth** button:



- The **Smooth** feature allows you to constrain a spline to be contiguous and maintain G2 continuity with another spline, line, arc, or polyline.
- NOTE** Before you apply the **Smooth** constraint you should first apply the **Coincident** constraint on the two curves.
- See the following illustration:



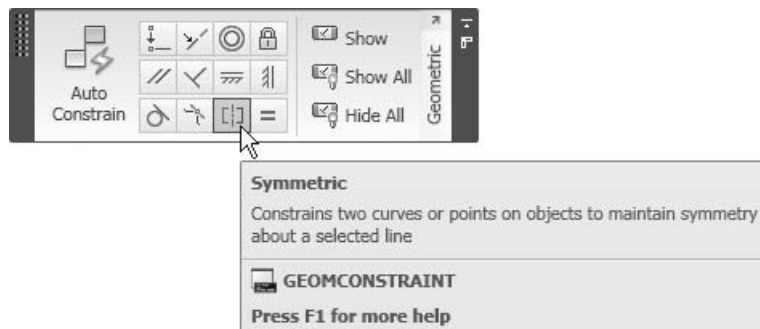
Before

Apply Coincident constraint

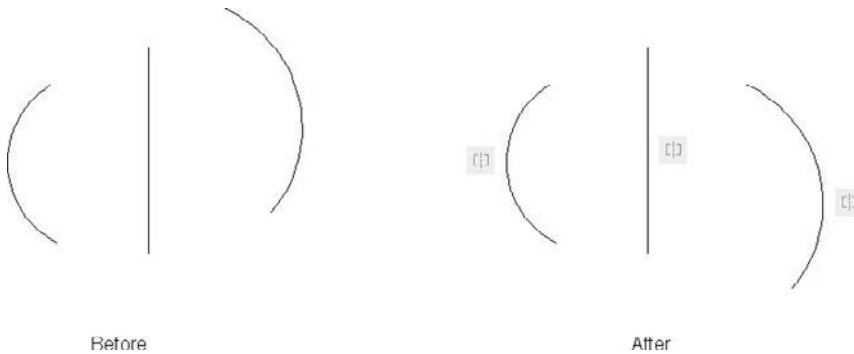
After

### Symmetric

- Using the **Geometric** panel, click the **Symmetric** button:

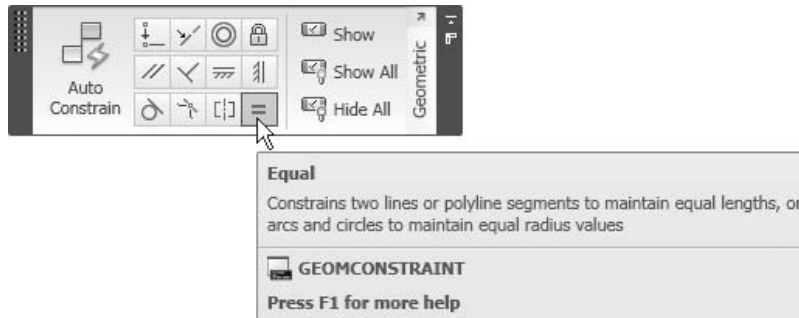


- The **Symmetric** function allows you to constrain two curves to maintain symmetry around a selected line.
- See the following illustration:



### Equal

- Using the **Geometric** panel, click the **Equal** button:



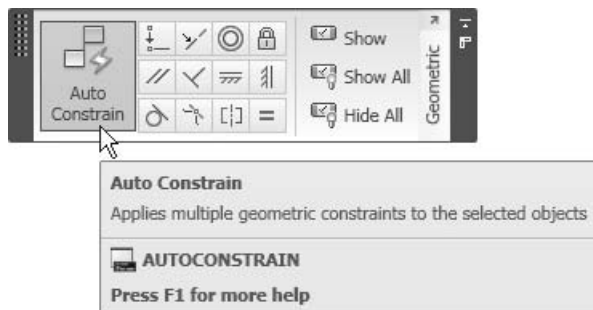
- The **Equal** feature allows you to constrain two lines or polyline segments to maintain equal lengths, or arcs and circles to maintain equal radius values.
- See the following illustration:



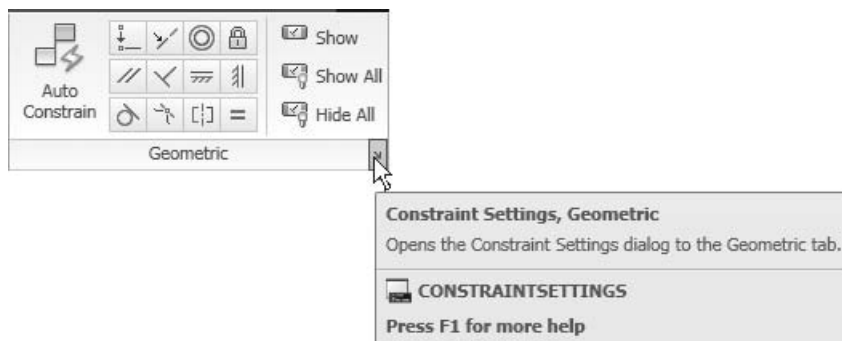


### Auto Constrain

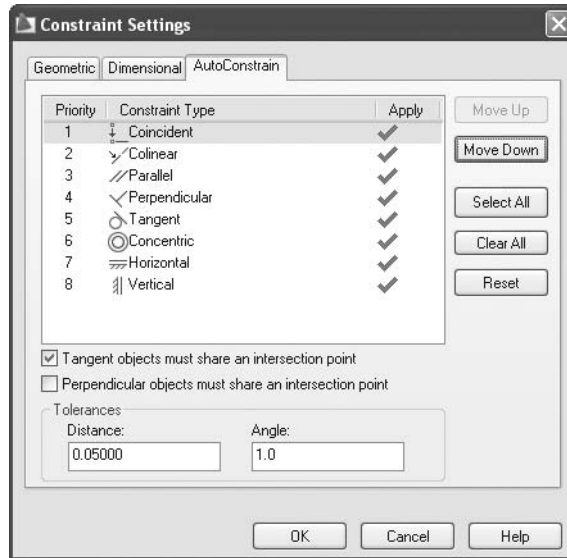
- Using the **Geometric** panel, click the **Auto Constrain** button:



- The **Auto Constrain** feature allows you to apply multiple geometric constraints to selected objects. It depends on two things:
  - The objects selected.
  - The order of the geometric constraints in the **Constraint Settings** dialog box, which you can reach through the **Auto Constrain** tab.
- You can reach the **Constraint Settings** dialog box by using the **Parametric** tab and clicking the arrow at the lower right-hand corner:



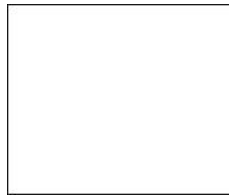
- You will see the following dialog box. Select the **AutoConstrain** tab:



- You can set the priority for each one of the constraints.
- You can set the rules for the **Tangent** and **Perpendicular** features.
- See the following example:

#### Example

- Begin with the following rectangle:



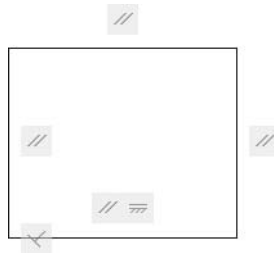
- Issue the **AutoConstrain** command. The following prompt will appear:

Select objects or [Settings]:

- Select the desired object (the rectangle), and then press [Enter]. The following message will appear:

8 constraint(s) applied to 4 object(s)

- You will now see the following picture:



- The eight constraints are: **Parallel** (2), **Horizontal**, **Perpendicular**, and **Coincident** (4).

### Show and Hide

- You can show or hide the geometric constraints using three functions:



- Using **Show**, you can show the constraints for some of the objects.
- Using **Show All**, constraints will be shown for all objects.
- Using **Hide All**, constraints will be hidden for all objects.

## GEOMETRIC CONSTRAINTS (GENERIC)



### Exercise 66

- Start AutoCAD 2010.
- Open the file *Exercise\_66.dwg*.
- Select the **Parametric** tab. Using the **Geometric** panel, select **Concentric**. Next, select the inner, larger circle as the first object and the outer, larger circle as second object.
- Try to move one of the two circles. What happens? \_\_\_\_\_
- Undo the **Move** command.
- Select the horizontal line and two outer vertical lines, and then click **Auto Constrain**. How many constraints were added? \_\_\_\_\_
- List the constraints that were added: \_\_\_\_\_
- Select **Equal** and the **Multiple** option. Select one of the small circles at the top, and then select the other circles.
- Save the file and close it.

### 13.3 DIMENSIONAL CONSTRAINTS

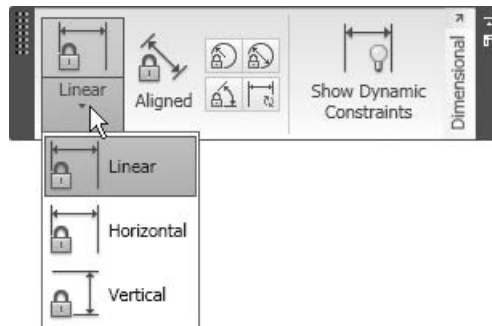
- In order to utilize these constraints, select the **Parametric** tab and use the **Dimensional** panel:



- There are eight different dimensional constraints, which are **Linear**, **Horizontal**, **Vertical**, **Aligned**, **Radial**, **Diameter**, **Angular**, and **Convert**.

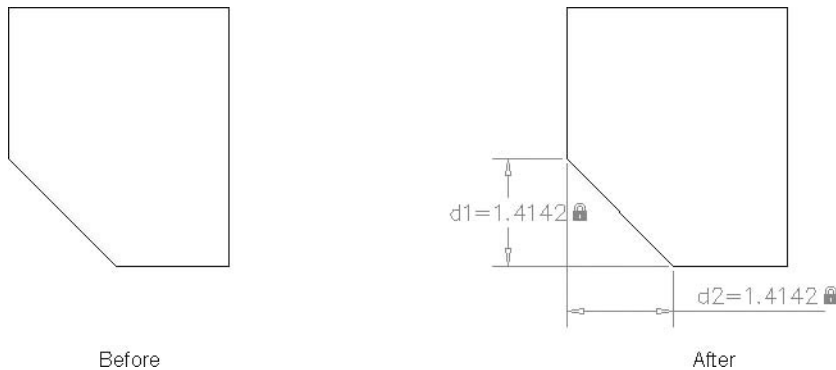
#### Linear, Horizontal, and Vertical

- The **Linear** button has three options:



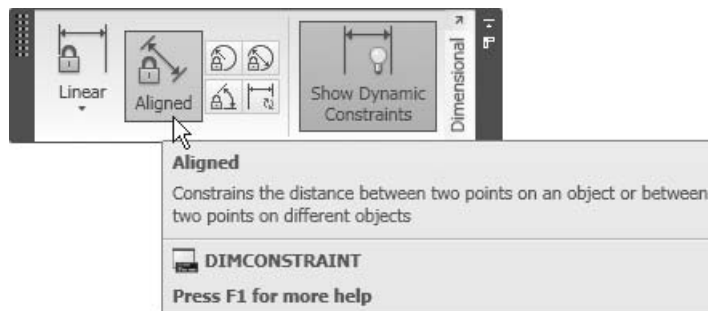
- The **Linear** option will constrain either horizontally or vertically.
- The **Horizontal** option will constrain the  $x$ -axis distance between two points.
- The **Vertical** option will constrain the  $y$ -axis distance between two points.

- See the following illustration:

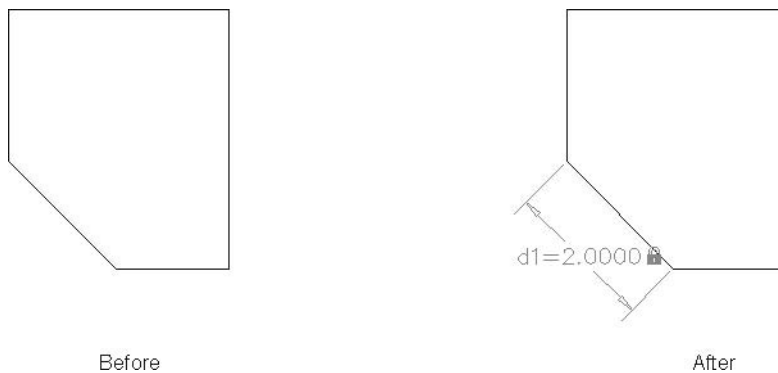


### Aligned

- Using the **Dimensional** panel, click the **Aligned** button:

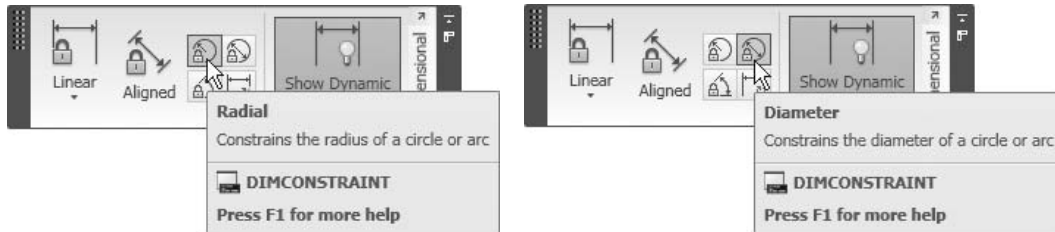


- The **Aligned** feature allows you to constrain the distance between two points.
- See the following illustration:

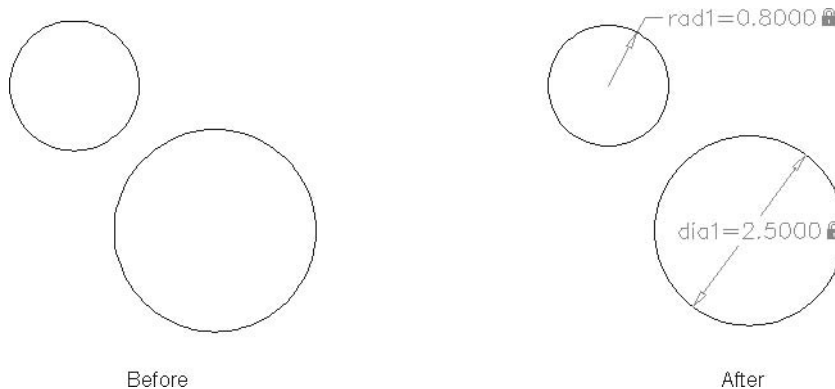


### Radial and Diameter

- Using the **Dimensional** panel, click the **Radial** or the **Diameter** button:

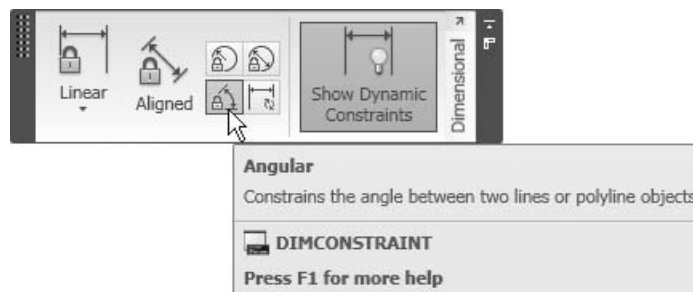


- The **Radial** and **Diameter** features allow you to constrain the radius or the diameter of a circle or an arc.
- See the following illustration:



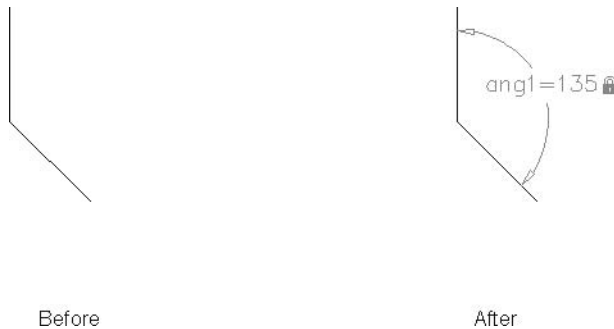
### Angular

- Using the **Dimensional** panel, click the **Angular** button:



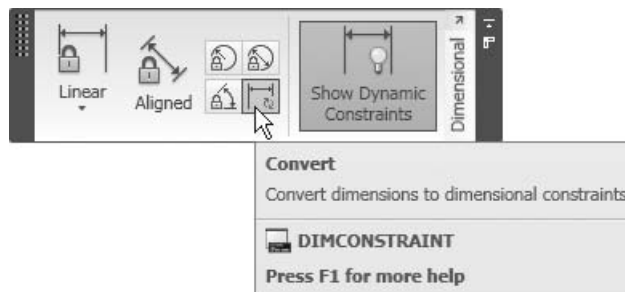
- The **Angular** feature allows you to constrain the angle between two lines or polyline objects.

- See the following illustration:

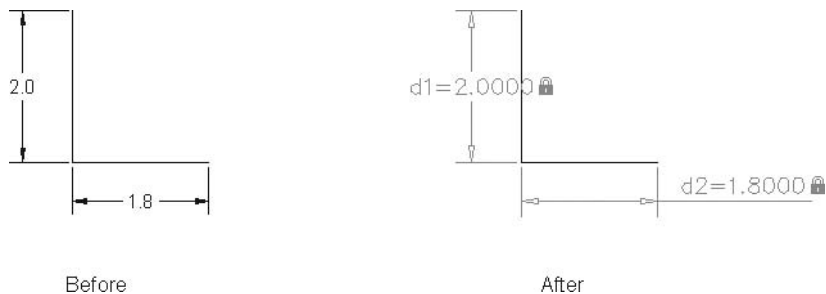


### Convert

- Using the **Dimensional** panel, click the **Convert** button:

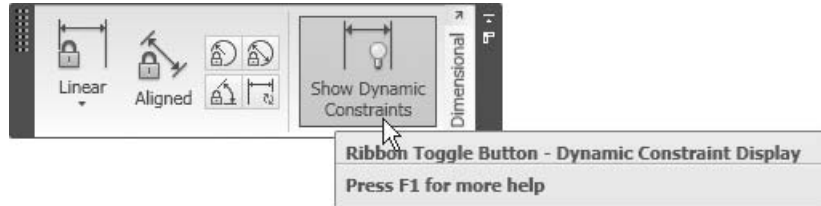


- The **Convert** feature allows you to convert dimensions to dimensional constraints.
- See the following illustration:

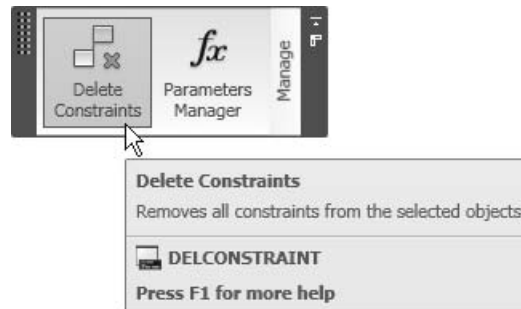


### Show Dynamic Constraints

- Using the **Dimensional** panel, click the **Show Dynamic Constraints** button:



- The **Show Dynamic Constraints** feature allows you to show or hide the **Dimensional Constraints**.
- NOTE** You can delete a constraint by selecting the **Parametric** tab and, using the **Manage** panel, clicking the **Delete Constraints** button. Next, select the desired objects.



## DIMENSIONAL CONSTRAINTS (GENERIC)



### Exercise 67

- Start AutoCAD 2010.
- Open the file *Exercise\_67.dwg*.
- Using the **Parametric** tab and the **Dimensional** panel, select **Horizontal**. Right-click and select **Object**, and then select the horizontal line. Accept the **current length** = 9.4 and press [Enter].
- Using the **Aligned** command, set the length of the small vertical line farthest to the right.
- Convert the existing dimension to be a constraint.
- Constrain the upper small circle to its current diameter.
- Save the file and close it.



## 13.4 DYNAMIC BLOCKS: AN INTRODUCTION

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- **Dynamic Blocks** were introduced for the first time in AutoCAD 2006.
- The **Dynamic Blocks** feature allows you to deal with blocks that have multiple features (i.e., different sizes, different views, and different shapes) that can be combined into a single block.
- Before AutoCAD 2006, the only types of blocks available were static blocks. With static blocks, you have only one shape, one size, and one view.
- The **Dynamic Blocks** feature offers the following benefits:
  - You will deal with a smaller number of blocks. Big companies that normally deal with thousands and thousands of static blocks will have the chance to downsize them to hundreds of blocks.
  - Fewer mistakes will be made in dealing with the different shapes and sizes of a certain block. For example, the **Door** block for size 1' will include the plan view and the elevation view; hence, you will always drag the same block whether you are in a plan drawing or in an elevation or section drawing. All you need to do is change the view of the block.
  - Intelligent features like the **Alignment** command are included. These features allow the user to align the block based on an existing object and the **Flip** command, which allows the user to create a mirror image of the block.
  - If we assume that blocks occupy approximately 30% of any drawing, then using **Dynamic Blocks** will accelerate the process of inserting blocks, thereby expediting the creation of the drawing.

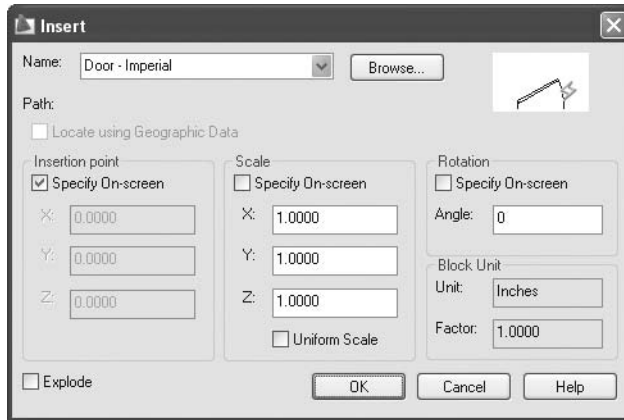
## 13.5 HOW DO YOU RECOGNIZE A DYNAMIC BLOCK?

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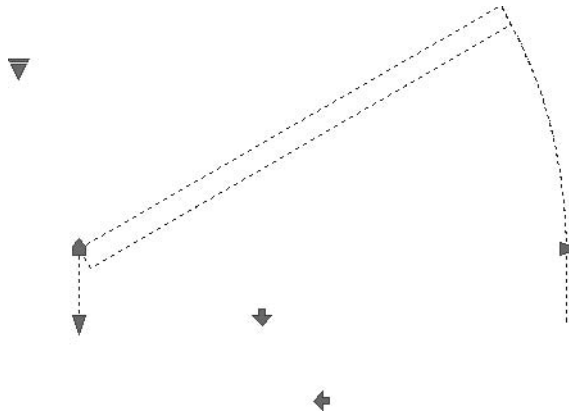
- If you are using **Tool Palettes**, you will see the following:



- If you are using the **Insert** command, you will see the following:



- If you clicked on a block, you will see something like the following:



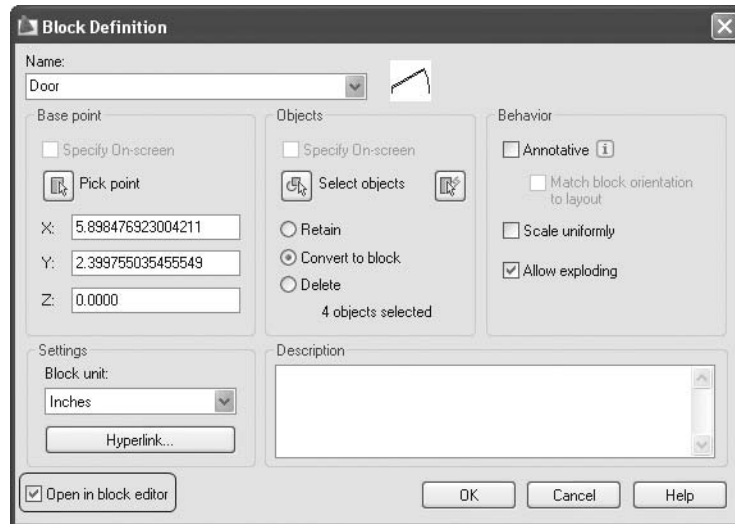
- Using all of these three methods, you can now know whether you are dealing with a dynamic block or not.

## 13.6 GETTING STARTED WITH DYNAMIC BLOCKS

- There are several ways to start creating a dynamic block in AutoCAD:
  - During the creation of a static block
  - Double-clicking an inserted block
  - Using the **Block Editor** command

### Creating a Static Block

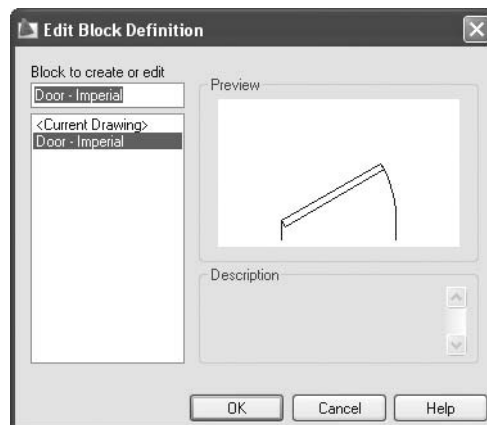
- When you want to create a static block, you must issue the **Block** command. The following dialog box will appear:



- You can see **Open in block editor** in the lower left-hand corner of this dialog box.
- If you check this checkbox, AutoCAD will create a static block and will open the **Block Editor**, giving you the ability to add the features to make it a dynamic block.

### Double-Clicking an Inserted Block

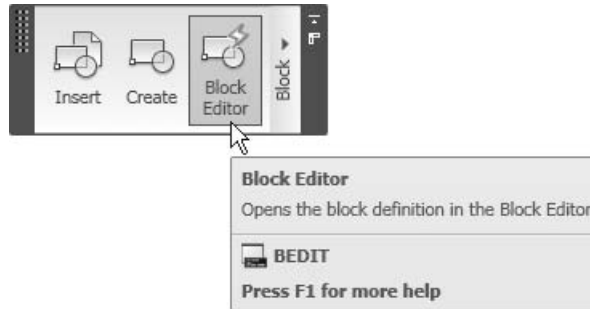
- If you double-click an existing block, the following dialog box will appear:



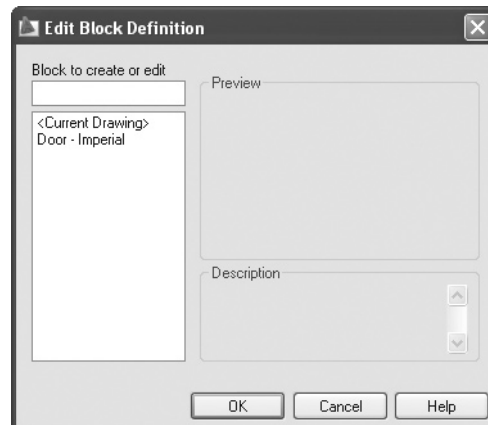
- In this example, we will open the **Door-Imperial** block in the block editor to add some dynamic features to it.

#### The Block Editor Command

- To issue this command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **Block** panel, click the **Block Editor** button:



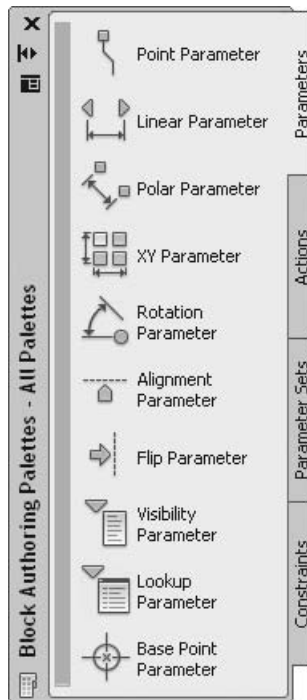
- You will see the following dialog box:



- Here you can select an existing block and edit it or you can create a new block from scratch.

## 13.7 THE AUTHORING TOOL PALETTE FOR DYNAMIC BLOCKS

- When you are inside the **Block Editor**, you will see a tool palette. It looks like the following:



- The **Block Authoring Palette** contains four tabs:
  - **Parameters**
  - **Actions**
  - **Parameter Sets**
  - **Constraints**

### Parameters

- A **Parameter** is a geometric definition of the block (e.g., **Linear**, **Polar**, **XY**, and **Rotation**).

### Actions

- **Actions** are commands implemented over a parameter. Almost all actions require you to select a predefined parameter.

### Parameter Sets

- **Parameter Sets** consist of both parameters and actions.

### Constraints



- These are the geometric and dimensional constraints discussed in the preceding sections. **Constraints** can be integrated inside a dynamic block.
- Each parameter and action has properties that can be modified.
- Properties of parameters and actions will play a pivotal role in controlling dynamic blocks.
- The following are the available **Parameters**:
  - A **Point Parameter** allows you to add a point definition to a dynamic block. **Stretch** and **Move** actions will utilize this parameter.



- The **Linear Parameter** allows you to add a linear dimension to the geometry of a dynamic block. This distance can have any angle. However, when you apply an action, users cannot change this angle. **Move**, **Stretch**, **Scale**, and **Array** actions will utilize this parameter.



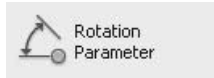
- The **Polar Parameter** is just like the **Linear Parameter**, except users will be allowed to change the angle while using this dynamic block. **Move**, **Stretch**, **Polar Stretch**, **Scale**, and **Array** actions will utilize this parameter.



- The **XY Parameter** is useful if you wish to define two associated dimensions (when the first changes, the second will also change). **Move**, **Stretch**, **Scale**, and **Array** actions will utilize this parameter.



- You can add a **Rotation Parameter** to the block definition. Only the **Rotate** action can utilize this parameter.



- The **Alignment Parameter** allows the block to align itself with any object in the drawing. No action utilizes this parameter.



- The **Flip Parameter** allows you to create a mirror image of a block around a selected axis. Only the **Flip** action utilizes this parameter.



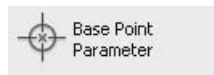
- The **Visibility Parameter** assists you in making several static blocks into a single dynamic block. You can choose which objects should be visible and which should be invisible. No actions utilize this parameter.



- The **Lookup Parameter** allows you to create tables of other parameters and arrange them as a list of views, sizes, and so on. Only the **Lookup** action utilizes this parameter.



- The **Base Point Parameter** allows you to define another base point for the block. No action utilizes this parameter.



- The following are the available actions:
  - The **Move Action** is similar to the **Move** command. You must select the parameter, specify the parameter point (base point), and then select the objects to be moved.



- The **Scale Action** is similar to the **Scale** command. You must select the parameter first and then select the objects to be scaled.



- The **Stretch Action** is similar to the **Stretch** command. You must select the parameter first, specify the parameter point (base point) next, then specify the stretch frame (the crossing concept in the **Stretch** command), and finally, select the objects to be stretched. **Stretch Action** will act only on the angle specified in the parameter.



- The **Polar Stretch Action** is identical to the **Stretch Action**, except it has an angle option.



- The **Rotate Action** is similar to the **Rotate** command. You must select the parameter first, and then select the objects to be rotated.





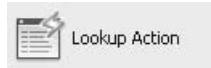
- The **Flip Action** is used to produce a mirror image around an axis specified in the **Flip Parameter**. You must select the parameter first, and then select the objects to be flipped.



- The **Array Action** is similar to the **Rectangular Array** command. You must select the parameter first, and then select the desired objects. If the **Linear Parameter** is selected, you will need to specify the number of columns. If the **XY Parameter** is selected, you will need to specify the number of rows and columns.



- The **Lookup Action** utilizes the **Lookup Parameter** to create a table of parameters and display them as a list for the users when they want to use the **Dynamic Block** command.



- The **Block Properties Table** creates and defines a new user parameter to be added to the **Properties Table**.



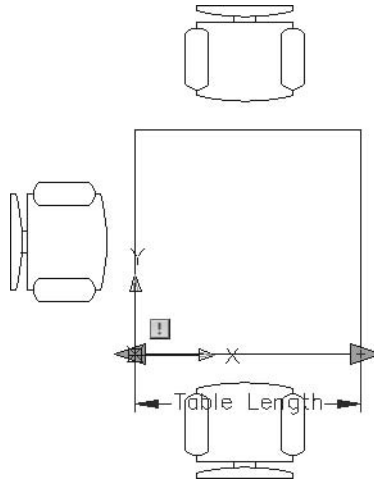
- **Parameter Sets** contain preset parameters and actions, with the following two exceptions:
  - **Pairs** require two actions to utilize one parameter. For example, the **Linear Stretch Pair** includes two **Stretch Actions** with one at each end of the **Linear Parameter**.
  - **Box Set** requires four actions to utilize one parameter. For example, the **XY Stretch Box Set** includes four **Stretch Actions** on each corner of the **XY Parameter**.



- In the **Block Editor** tab you will see a panel called **Actions Parameters**. This panel includes all the parameters and actions that we have just discussed.



- You can rename any parameter and action as you wish. See the following example:



- To rename a parameter or action, you must:
  - Click on any parameter or action, right-click, and select **Properties**.
  - Under **Property Labels**, select the **Distance Name** (as an example) and input the new name.

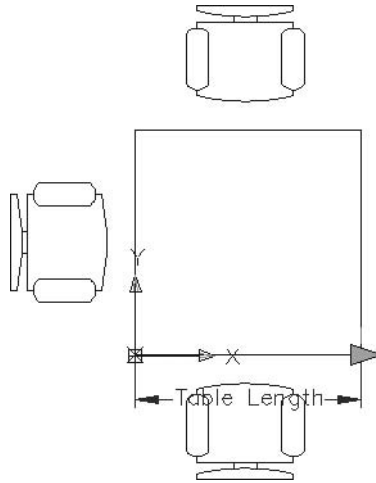
## DYNAMIC BLOCKS: PART 1 (GENERIC)



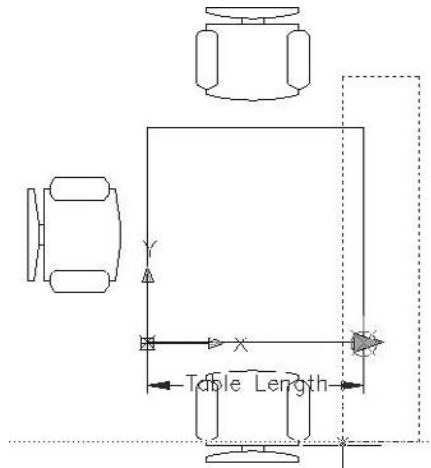
### Exercise 68

- Start AutoCAD 2010.
- Open the file *Exercise\_68.dwg*.
- You will find a table with three chairs.
- Issue the **Block Command** to define a new block using the following information:
  - **Block name** = Meeting Table
  - **Base point** = Lower left corner of the table
  - **Objects** = Select all objects and select **Delete** to delete the objects after creating the block.
  - Make sure that the **Open in block editor** option is turned on.
- Now you are inside the **Block Editor**.
- Using the **Block Authoring Tool Palettes**, select the **Parameters** tab, and then select the **Linear Parameter**. Select two points representing the length of the table (the lower horizontal line). Make sure to select the left point first and the right point second.

- Select the **Linear Parameter**, right-click, and select **Properties**.
- Change the **Distance** label to **Table Length**.
- Under **Misc** change **Number of Grips** to 1. (We make this change to prevent the table from being stretched to the left, which was also the reason for choosing the points from left to right.)



- Using the **Block Authoring Tool Palettes**, select the **Actions** tab, and then select the **Stretch Action**:
  - Select the defined parameter.
  - Select the lower right corner to associate this point with the action.
  - Since we know how to use the normal **Stretch** command, select the **Crossing Window**, which will stretch the table to the right side. See the following illustration:



- When you are prompted to select objects, specify the same **Crossing Window** (avoid selecting the chairs).
- The command will end.
- You do not want to leave it to the user to specify the length of the stretch. Therefore, select the parameter again, right-click, then select **Properties**. Specify the following under the **Value Set**:
  - **Dist type** = Increment
  - **Dist increment** = 40
  - **Dist minimum** = 40
  - **Dist maximum** = 120
- Using the **Block Authoring Tool Palettes**, select the **Actions** tab, and then select the **Array Action**:
  - Select the defined parameter.
  - Select the two chairs.
  - **Distance between columns** = 40
- Save the block.
- Click **Close Block Editor**.
- Insert the block **Meeting Table** and test it.
- Save the file and close it.

## DYNAMIC BLOCKS: PART 2 (IMPERIAL)

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### Exercise 69

- Start AutoCAD 2010.
- Open the file *Exercise\_69.dwg*.
- You will find an empty file.
- Start the **Block Editor** command and create a new block named **Chairs and Sofas**.
- Insert the following existing blocks specifying the insertion point as 0,0 for each.
  - **Chair-Desk**
  - **Chair-Rocking**
  - **Sofa Roundback 7 ft**
  - **Sofa Roundback Loveseat 5 ft**
- Double-click to see all blocks.
- Although there are four blocks on top of each other, it is very easy to select just one of them.
- Turn the **Visibility Mode** button on.
- Using the **Block Authoring Tool Palette** and the **Parameters** tab, click the **Visibility Parameter** and insert it in a suitable place.

- Using the **Visibility** panel, click the **Visibility States** button and rename the existing state as **Chair–Desk**.
- Using the **Visibility** panel again, click the **Make Visible** button and click the block **Chair–Desk**. Click the **Make Invisible** button and make the other three blocks invisible—only in the current visibility state.
- You can click the button **Visibility Mode** to gray out the blocks rather than making them totally invisible.
- Using the **Visibility** panel, click the **Visibility States** button and **New**. Create a new state as **Chair–Rocking**.
- Clicking the **Make Visible** button, make the **Chair–Rocking** visible. Clicking the **Make Invisible** button, make the **Chair–Desk** invisible.
- Repeat the same procedure for the other two sofas.
- Test your block before saving it and close it using the small list in the panel.
- Save and close the **Block Editor**.
- Insert the new block and test it.
- Save the file and close it.

## DYNAMIC BLOCKS: PART 3 (IMPERIAL)

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### Exercise 70

- Start AutoCAD 2010.
- Open the file *Exercise\_70.dwg*.
- You will find a preexisting block called **Window**.
- Double-click the block, and then select block **Window**. Click **OK** and the **Block Editor** will open.
- Add an **XY Parameter**, selecting the lower left corner of the shape, and the upper right corner.
- Add a **Scale Action** to the **XY Parameter**, selecting all objects.
- Click the **XY Parameter**. Right-click and select **Properties**. Specify the following:
  - **Number of Grips** = 0
  - Under **Value Set**, set **Hor type** to **List**, and input the new values 3'–6", 4'–6".
  - Set the **Ver type** to **List** and input the new values 1'–9", 2'–3".
- Add a **Lookup Parameter** and **Lookup Action**.
- In the **Lookup Action**, click the **Add Properties** button and select the two parameters available.
- In the table, add the proper Y value for each X value. Add a suitable lookup value, like **Window 3'–6" x 1'–9"**, for example.

- Complete the values for each of the three windows.
- Save and close the **Block Editor**.
- Test the block.
- Save the file and close it.

## **13.8 BLOCK ATTRIBUTES: AN INTRODUCTION**

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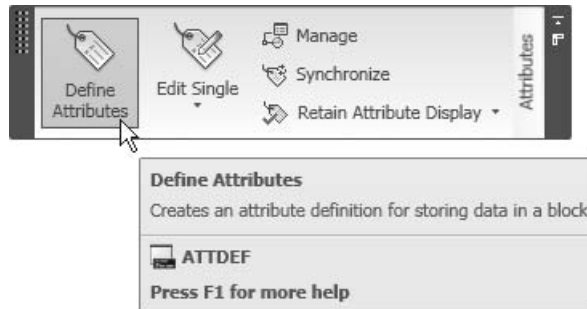
- **Block Attributes** is a method of adding information inside a block.
- There is no new command to create a block with attributes. You will still use the **Block** command. The only difference is that when you are finished drafting the desired shape, you can add the attributes inside or outside that shape and use the **Block** command to define the new block.
- There are two phases for **Block Attributes**.
  - Phase I: Create the attribute definitions. This phase will take place while you are defining the block. At this time, attributes will be considered as queries to the user (e.g., **Door number**, **Door Material**, **Door Model**).
  - Phase II: Insert blocks with attributes. The user will input the values (answers) of the attributes for this specific block (e.g., **D12**, **Wood**, and **DW-01-201**).
- Users can control the visibility of the attributes with the blocks; they can show them or hide them as they wish.
- Once you have added the attribute values to the different blocks inserted in your drawing, you will have two options available for editing:
  - Modify the values of the attributes.
  - Modify the attribute definitions themselves.
- If all of the data was added correctly to the drawing, you can extract this data either as a table in the drawing or export it to Microsoft® Excel® or any database engine.

## **13.9 DEFINING BLOCK ATTRIBUTES**

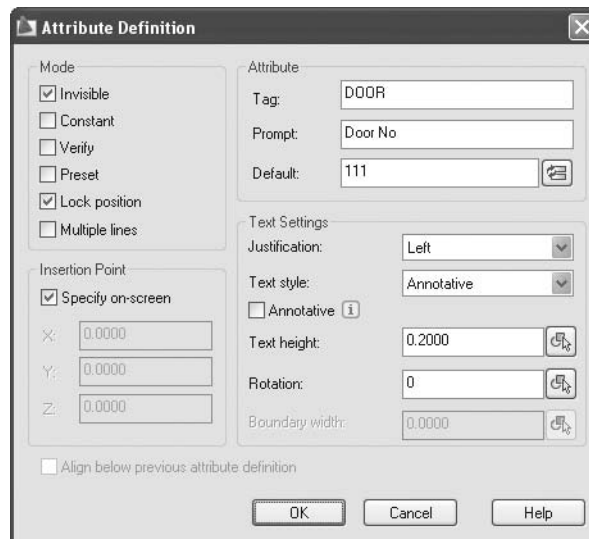
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- The first step in the attribute workflow is to create the attributes.
- Creating text style(s) prior to creating the attributes is highly recommended.

- To access this command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **Attributes** panel, click the **Define Attributes** button:



- You will see the following dialog box:



### Tag

- **Tag** is the name of the attribute. If you are familiar with any programming language, this is identical to a *variable* (e.g., **DOORMAT**).

### Prompt

- **Prompt** is the query that will appear after the user inserts the block asking for the data (optional). **Door Material** is an example.

### Default



- **Default** is the value that will appear to users (optional). **Wood** is an example.
  - Thus, the insertion of a field can be the default value of the attribute.
  - This practice is the only way to link stored data in AutoCAD to attributes.

### Invisible

- The **Invisible** option allows you to create an invisible attribute. Once the block is inserted the attribute value will not appear.

### Constant

- The **Constant** option allows you to create a constant value for all of the insertions.

### Verify

- The **Verify** option will prompt the user to enter the value twice. (This option is only applicable with no dialog box input.)

### Preset

- The **Preset** option means that the attribute will equal the default value.

### Lock Position

- The **Lock Position** option allows you to lock the position of the attribute reference to the block geometry. If it was off, you can change the attribute position later.

### Multiple Lines

- By default, the attribute is a single line text. However, with attributes like **Address**, you may need to input multiple lines. The **Multiple Lines** option will allow you to do so.

### Justification

- The **Justification** option allows you to justify the text reference to the insertion point. This option is identical to the single line text **Justification** option.



**Text Style**

- The **Text Style** option allows you to choose from preexisting text styles that will dictate the characteristics of the attribute text.

**Text Height**

- If the text style specified in the preceding step has not specified a value for the height, you can use the **Text Height** option to do so. Otherwise, the height will default to the height specified in the text style.

**Rotation**

- The **Rotation** option allows you to control the rotation angle of the attribute text.

**Boundary Width**

- The **Boundary Width** option is available only when the **Multiple Lines** option is on. This option will specify the maximum line length for multiple lines before the editor wraps the text.

**Insertion Point**

- The **Insertion Point** option allows you to specify the location of the attribute text reference in the block geometry. You have two choices, either type in the **X, Y, Z** of the attribute location, or choose **Specify on-screen**.



- If you have already inserted one attribute and you want the second one to be aligned below the first one, then select the following option:

☒ Align below previous attribute definition

- Once you turn this option on, all the text settings and insertion points will be dimmed to indicate that they are all preset.



- After you define the attributes near or within the objects that will be blocks, use the **Block** command to define the block with attributes. It is preferable to select the attributes in the same order in which you wish to show the user.

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**DEFINING ATTRIBUTES (GENERIC)**

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**Exercise 71**

- Start AutoCAD 2010.
- Open the file *Exercise\_71.dwg*.
- Make current layer = **Text**.

- Zoom to the door on the right side.
- Define the following parameters:

|                        |                             |                   |                                |
|------------------------|-----------------------------|-------------------|--------------------------------|
| <b>Tag</b>             | DOORNO                      | DOORH             | DOORW                          |
| <b>Prompt</b>          | Door Number                 | Door Height       | Door Width                     |
| <b>Default</b>         | —                           | 2.00              | Field                          |
| <b>Mode</b>            | —                           | Invisible, Preset | Invisible                      |
| <b>Text Style</b>      | Arial_010                   | Arial_010         | Arial_010                      |
| <b>Justification</b>   | Middle Center               | Left              | Left                           |
| <b>Insertion Point</b> | At the center of the circle | Below the Door    | Align below previous attribute |

- The **Field** information to be specified:
  - Object
  - Select the outside vertical line
  - Select **Length**
  - **Decimal** = 0.00
- Define a block from the shape and attribute. Define the following:
  - Block name = **Door – New**
  - **Insertion point** = Lower left corner of the door
  - **Select objects** = Select all objects first without the attributes. Next, select the attributes from top to bottom, one-by-one, then select **Delete**.
- Save the file and close it.

### 13.10 INSERTING BLOCKS WITH ATTRIBUTES

- The first step to insert a block with attributes is to choose the way the attribute definitions will be displayed so that you can fill in the values.
- System variable **ATTDDIA** will control this issue:
  - If **ATTDDIA** = 0, the attribute definitions will be shown in the **Command Window**.
  - If **ATTDDIA** = 1, a dialog box will appear to fill in the data.

- If you select the second choice and insert a block with attributes, the following dialog box will appear:

**Edit Attributes**

Block name: Door Type - 01

Door Number: 121

Door Height: 90

Door Width: 05

Door Material: Wood

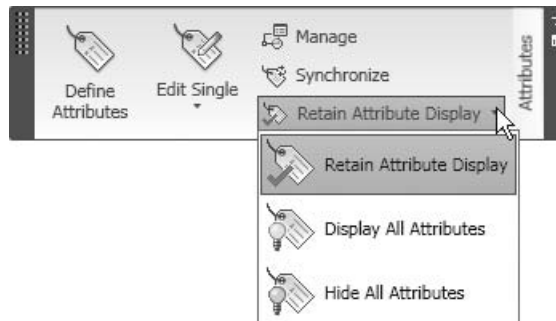
Door Manufacturer: IKEA

Cost: 295

OK Cancel Previous Next Help

## 13.11 BLOCK ATTRIBUTES: CONTROLLING THE VISIBILITY

- There are three visibility modes for attributes:
  - **Normal** means that all attributes will be “as defined.” For example, if the visibility mode was defined as invisible, the attributes will be invisible.
  - **On** means that all attributes will be visible regardless of their original definitions.
  - **Off** means that all attributes will be invisible regardless of their original definitions.
- To issue any of these three options, make sure you are on the **Insert** tab on the **Ribbon** and, using the **Attributes** panel, click the **Visibility** button:



- You can also control the visibility using the system variable **ATTMODE**: 0 = **Off**, 1 = **Normal**, and 2 = **On**.

## INSERTING BLOCKS WITH ATTRIBUTES (GENERIC)

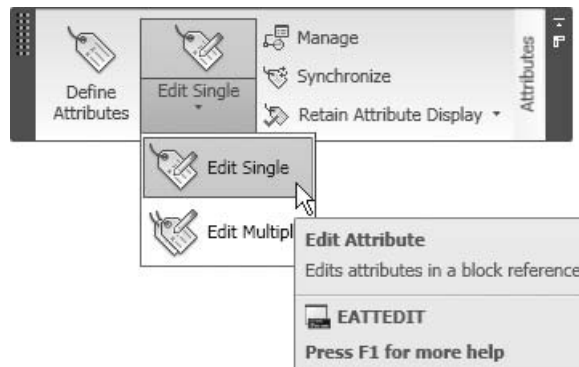


### Exercise 72

- Start AutoCAD 2010.
- Open the file *Exercise\_72.dwg*.
- Make current layer = **Door**.
- As you can see, there are three openings: from left to right, 1.00, 0.90, and 0.80.
- Insert block **Door – New**, assigning the following values:
  - **Door Number** = 01 for all three blocks
  - For the first opening, **X Scale** = 1.0
  - For the second opening, **X Scale** = 0.9, **Scale Uniform** = on
  - For the third opening, **X Scale** = 0.8, **Scale Uniform** = on
  - Notice how the **Door Width** will change automatically, as the default value is a *field*.
- Using **Attribute Visibility**, adjust the visibility as you wish.
- Save the file and close it.

## 13.12 BLOCK ATTRIBUTES: EDITING VALUES (ONE-BY-ONE)

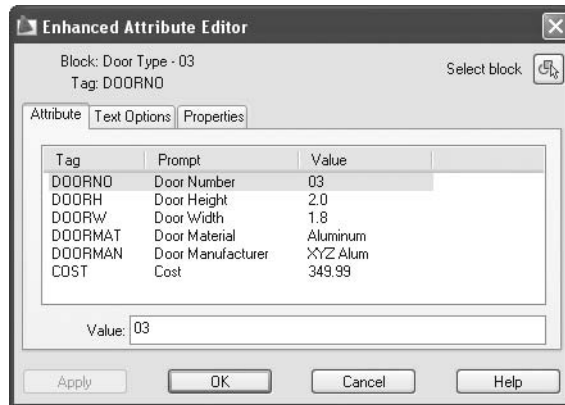
- After inserting blocks with attributes, you may need to edit the values (we are not talking about the attribute definitions).
- You can edit the values of a single block one at a time.
- To issue this command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **Attributes** panel, click the **Edit Single** button:



- The following prompt will appear:

Select a block: (*Select the desired block with attributes*)

- Once you select the desired block, the following dialog box will appear:



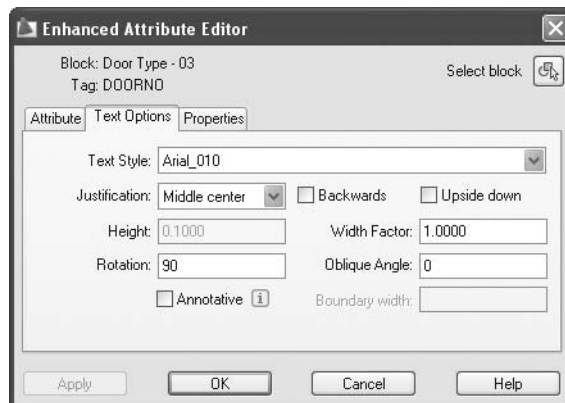
- At the top of the dialog box you will see the name of the block selected.
- There are three tabs in this dialog box:
  - **Attribute**
  - **Text Options**
  - **Properties**

### Attribute

- In this tab, you will see a list of the attributes and their respective values. You can edit any or all of the values.
- Select the desired attribute and input the new value in the field **Value**.

### Text Options

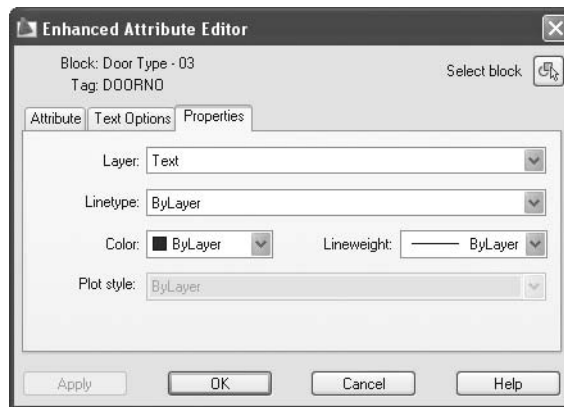
- If you click on this tab, you will see the following dialog box:



- You can change all or any of the following (all of these preexist in the **Text Options** dialog box):
  - **Text Style**
  - **Justification**
  - **Backwards and Upside down**
  - **Height**
  - **Width Factor**
  - **Rotation Angle**
  - **Oblique Angle**
  - **Boundary width** (for multiple line attributes)
  - **Annotative**

### Properties

- If you click on this tab, you will see the following dialog box:



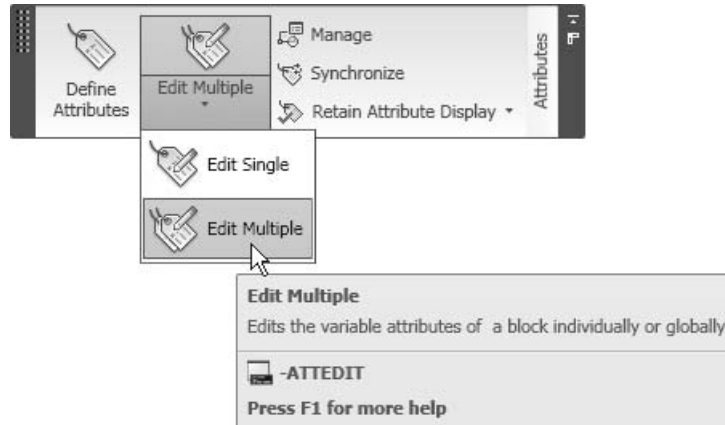
- You can change any or all of the following:
  - **Layer**
  - **Linetype**
  - **Color**
  - **Lineweight**
  - **Plot style**



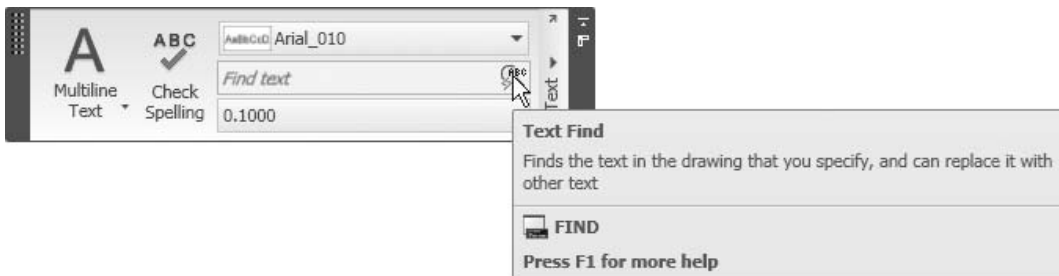
- Another way to reach the **Enhanced Attribute Editor** is to double-click the block.

## 13.13 BLOCK ATTRIBUTES: EDITING VALUES (GLOBAL)

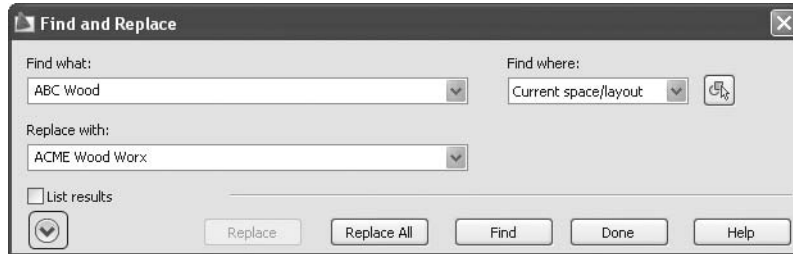
- Make sure you are on the **Insert** tab on the **Ribbon** and, using the **Attributes** panel, click the **Edit Multiple** button:



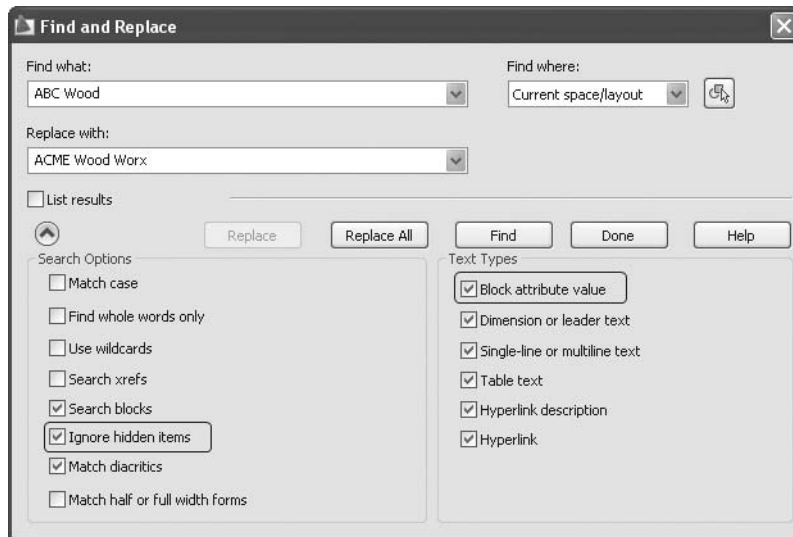
- The **Edit Multiple** command allows you to edit the values of attributes for all blocks in the same drawing.
- This command is somewhat lengthy and tedious. It is highly recommended that you use the **Find** command instead because it is straightforward and will do the same function in fewer steps. Also, the same command is used in other software packages, thus it is more widely known.
- In order to issue this command, make sure you are on the **Annotate** tab on the **Ribbon** and, using the **Text** panel, click the **Text Find** button:



- When you click the **Text Find** button, the following dialog box will appear:



- If you click the small button in the lower left corner, the following dialog box will appear:



- In the **Find what** field, type in the current value of the text you wish to change. In the **Replace with** field, type in the new value. You can also specify whether to search for this string in either the **Entire drawing** or in a specified window.
- Be sure to select the checkbox **Block attribute value**.
- Select one of the following choices:
  - **Find** allows you to find the text only without replacing it.
  - **Replace** allows you to find and replace the text one-by-one.
  - **Replace All** allows you to replace all text in this drawing at once.
- If your desired attributes are invisible, then turn **Ignore hidden items** off.



## EDITING ATTRIBUTE VALUES (GENERIC)

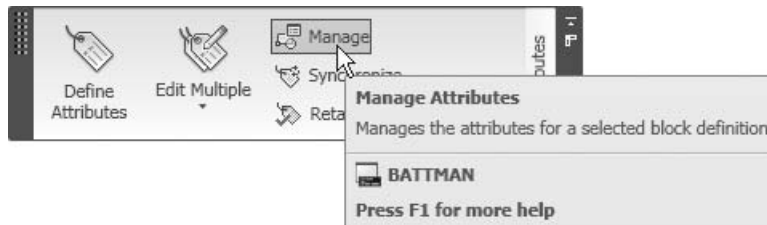


### Exercise 73

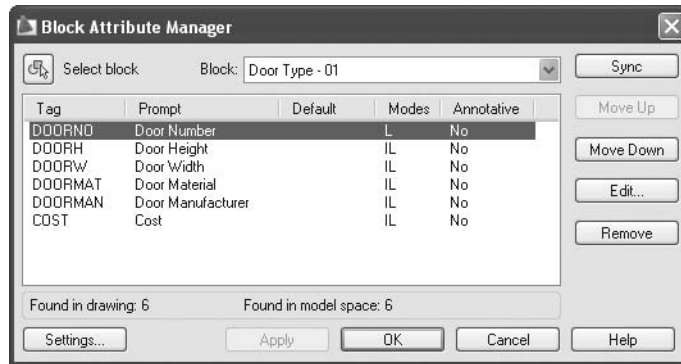
- Start AutoCAD 2010.
- Open the file *Exercise\_73.dwg*.
- Using the **Find** command, perform the following steps:
  - Make sure that **Ignore hidden items** is turned off.
  - Replace **ABC Wood** with **Acme Wood Works**.
  - Replace **199.99** with **149.99**.
- Select any door, double-click it, and change the color of the **DOORNO** to cyan.
- Select another door, double-click it, select the **DOORNO** attribute, and make the **Text style** = ISO Proportional, and **Height** = 0.2.
- Save the file and close it.

## 13.14 REDEFINING ATTRIBUTES

- The **Redefining Attributes** command will help you change the attribute definitions (not the values).
- This command will not help you add new attributes to an existing block. If you wish to add a new attribute to an existing block, you have to explode it first, define a new attribute, and then redefine it under the same name.
- In order to issue this command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **Attributes** panel, click the **Manage** button:



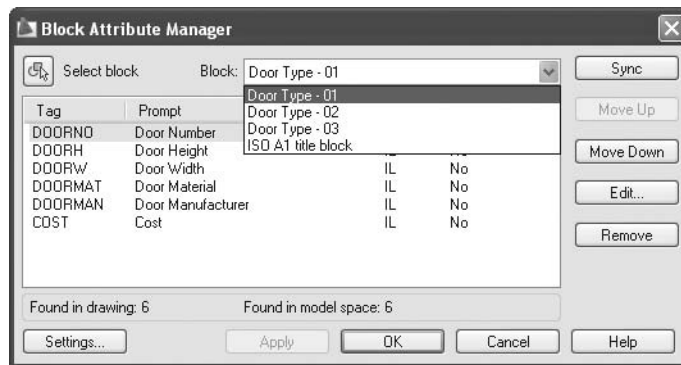
- The following dialog box will appear:



- This command is used to change the definition of an existing attribute, *not* the value of this attribute.
- As a first step, you need to select an existing block. There are two ways to do that:



- If you don't know the name of the block, click the button at the top left side of the dialog box and select the desired block.
- If you know the name of the desired block, select the block name from the list beside the button as shown:



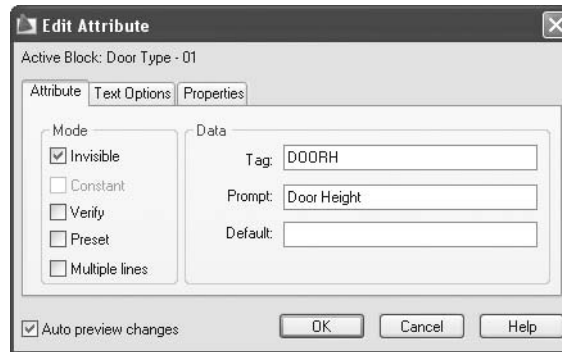
- Accordingly, a list of the attributes associated with this block will appear.
- Next, you must select the desired attribute in order to make one of the following modifications:



- This modification allows you to move the selected attribute one position up.
- This modification allows you to move the selected attribute one position down.
- This modification allows you to delete the selected attribute.



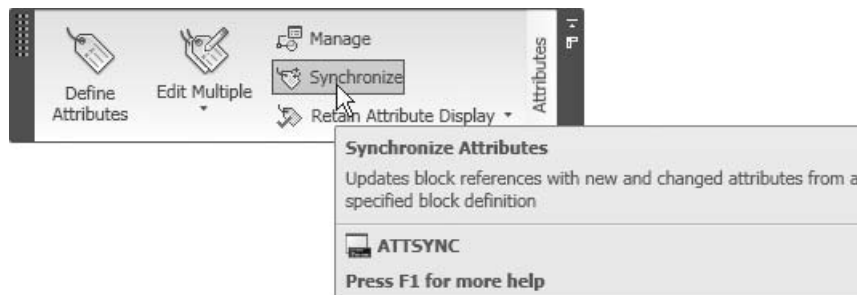
- This modification allows you to edit the selected attribute. When you click **Edit**, the following dialog box will appear:



- You can edit all attribute-defined information like **Tag**, **Prompt**, **Default value**, or **Mode**.
- **Text Options** and **Properties** have already been covered in this text.

## 13.15 THE SYNCHRONIZE COMMAND

- If the user exploded a block with attributes, added a new attribute, and then redefined the new block with the same name, all the new insertions will show the new attribute(s). What about the already-inserted blocks?
- The **Synchronize** command will update all of the existing attributes and any new attribute that has been added.
- Another version of this command is issued through the **Block Attribute Manager** button **Sync**.
- In order to issue the **Synchronize** command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **Attributes** panel, click the **Synchronize** button:



- The following prompt will appear:

Enter an option [/?/Name/Select] <Select>:

- If you memorize the spelling of the block name, type **n** and press [Enter]. The following prompt will appear:

Enter name of block to sync or [?]:

- Type the name of the block and the block will be synchronized instantly.
- If you don't recall the exact spelling of the block name, then type **s** and press [Enter]. The following prompt will appear:

Select a block:

- Select the desired block and you will see the following prompt:

ATTSYNC block Door? [Yes/No] <Yes>:

- To make sure that you didn't make a mistake, AutoCAD will display the name of the selected block (in our example, the name of the block is **Door**). You will be prompted to confirm either Yes or No. AutoCAD will respond accordingly.
- In the **Block Attribute Manager** dialog box, you can see the **Sync** button at the upper right corner. This button will do exactly what the **Synchronize** command does.



- However, this button will not help if you are adding new attributes.

## **REDEFINING AND SYNCHRONIZING ATTRIBUTES (GENERIC)**



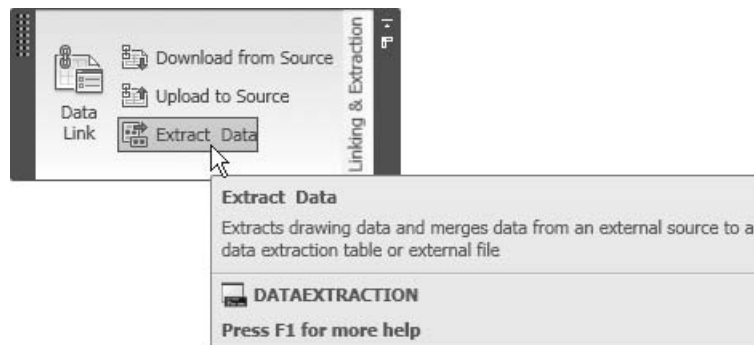
### **Exercise 74**

- Start AutoCAD 2010.
- Open the file *Exercise\_74.dwg*.
- Make the current layer = **Door**.
- Insert the block **Door Type - 01** in an empty space and explode it.
- Make the current layer = **Text**.
- All of the defined attributes will appear.
- Add a new attribute. Insert it below the last existing attribute with the following information:
  - **Tag** = MANADD
  - **Prompt** = Manufacturer Address
  - **Mode** = Invisible, Multiple lines

- Define the block again under the same name (while defining the block, don't forget to select the objects first, then the attributes in the right sequence). Answer **Yes** if AutoCAD displays the message: "This block exists, do you want to redefine it?"
- Use the **Synchronize** command to make the other **Door – Type 01**. View the changes.
- Double-click on any of these doors and make sure that a new attribute has been added.
- Use the **Block Attribute Manager** to make the following changes to **Door – Type 03**:
  - Make the **Cost** attribute the second in the list.
  - Make the **DOORH** attribute **Preset**, with **Default** value = 2.00.
- Insert **Door – Type 03** in an empty space and check the attributes. Can you see the new changes you made?
- Save the file and close it.

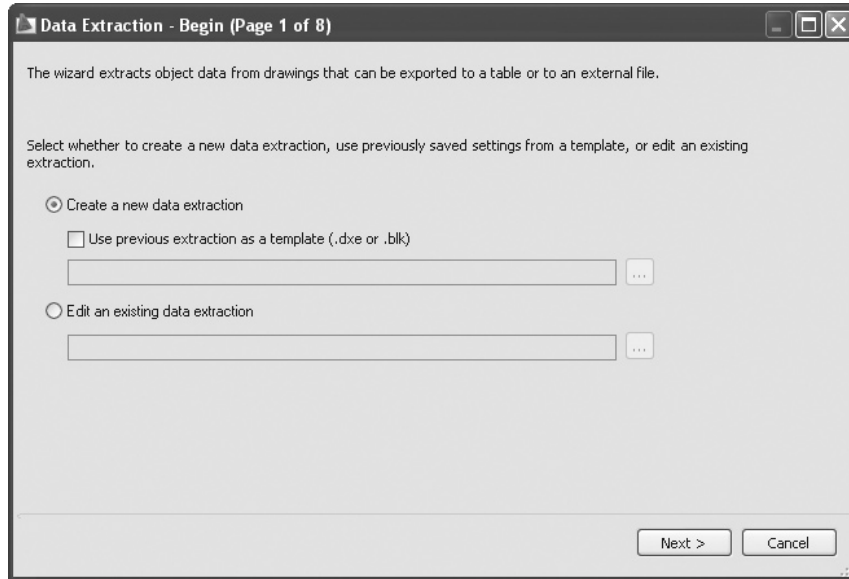
## 13.16 EXTRACTING ATTRIBUTES

- After you are done with data entry for all of your blocks and all of your attributes for the current drawing or group of drawings, you can extract these values and arrange them using a table that can be inserted in an AutoCAD drawing, an Excel spreadsheet, or a database engine.
- Use the **Data Extraction** command to extract attributes from drawing file(s).
- The **Data Extraction** command is a wizard that will assist the user step-by-step. To issue the command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **Linking & Extraction** panel, click the **Extract Data** button:

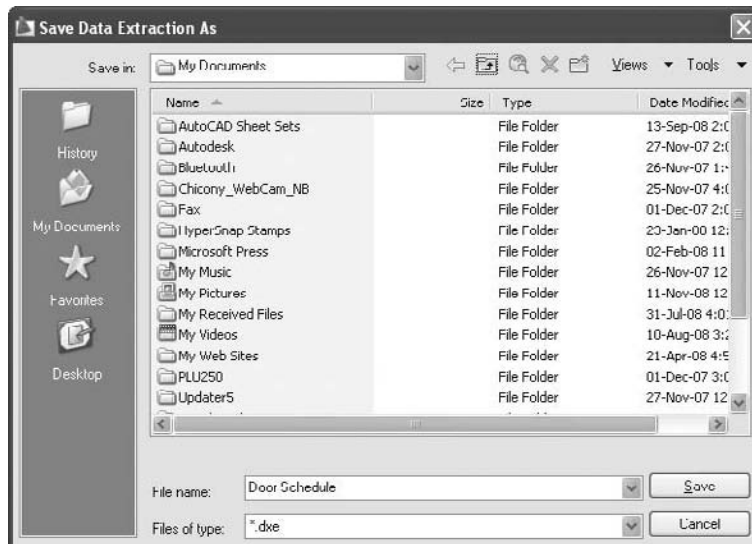


### Step One

- You will see the following dialog box:



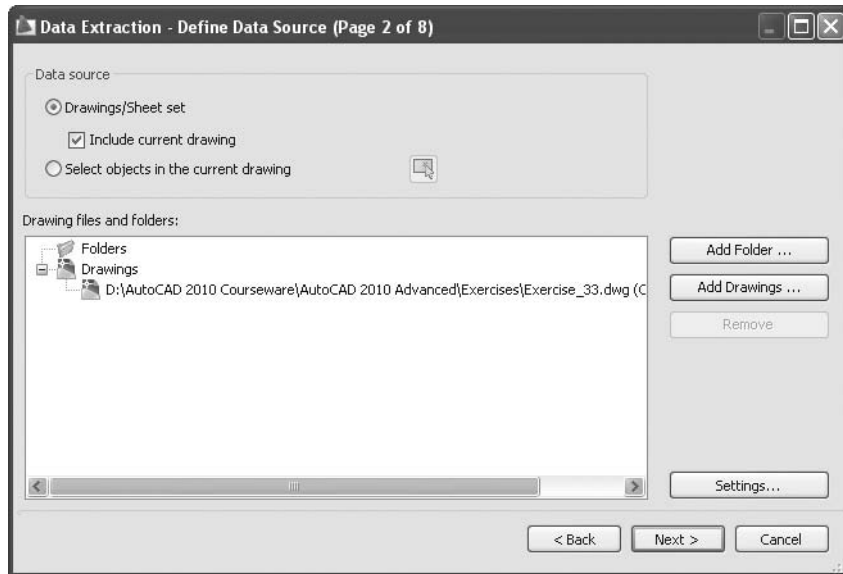
- You can do one of three things in this dialog box:
  - Create a new data extraction.
  - Use a previous extraction as a template (this will save time, if there are similar extractions).
  - Edit an existing data extraction.
- The **Save Data Extraction As** dialog box will appear:



- As you can see, the file extension will be *\*.dxe*. Type the name and click **Save** and **Next**.

### Step Two

- The following dialog box will appear:



- Now you must tell AutoCAD where it should extract the data:
  - From objects in the current drawing file
  - From the entire current file
  - From a group of drawing files
- By default, you will extract all data from all blocks in the current file.
- If you only want part of the current file, you must choose the option **Select objects** in the current drawing and then click the small button on the right. The dialog box will disappear temporarily, giving you the chance to select the desired blocks.
- Click the **Add Drawings** button to select files from different drives and different folders.



Add Drawings ...

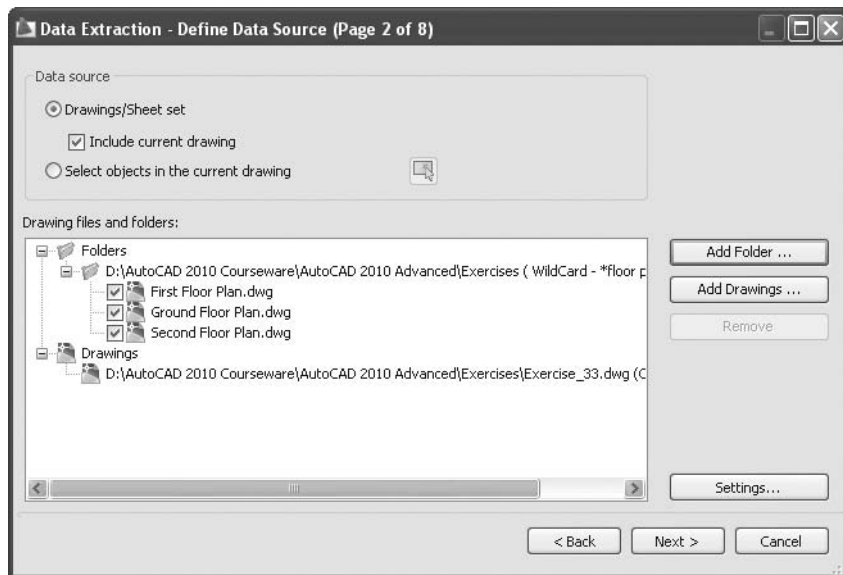
- Click the **Add Folder** button to see the following dialog box:



- In this dialog box you will specify a folder that contains all the drawings you need (you have to be careful to include any unwanted drawings in the desired folder).
- You have three options to control. They are:
  - **Automatically include new drawings added in this folder to the data extraction.** This option gives you the ability to include future drawing files in the extraction. This will keep the extraction process up-to-date.
  - **Include subfolders** offers you the choice to include any subfolders within the specified folder.
  - **Utilize wild-card characters to select drawings.** If you wish to have only some of the files in the folder rather than all of them, you can if you type a wild card, such as *c\*.dwg*, which means all drawing files (in this folder) starting with the letter **c** will be included.
  - Click **OK** once you are done.
- Based on your choices, you will see the list of the files matching your selection. See the following dialog box.



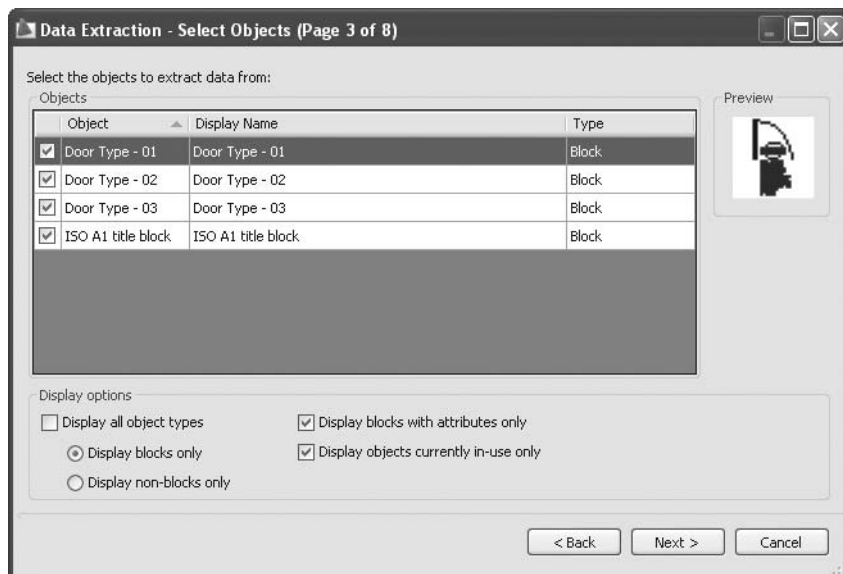
- This is an example of what you may have:



- Click **Next**.

### Step Three

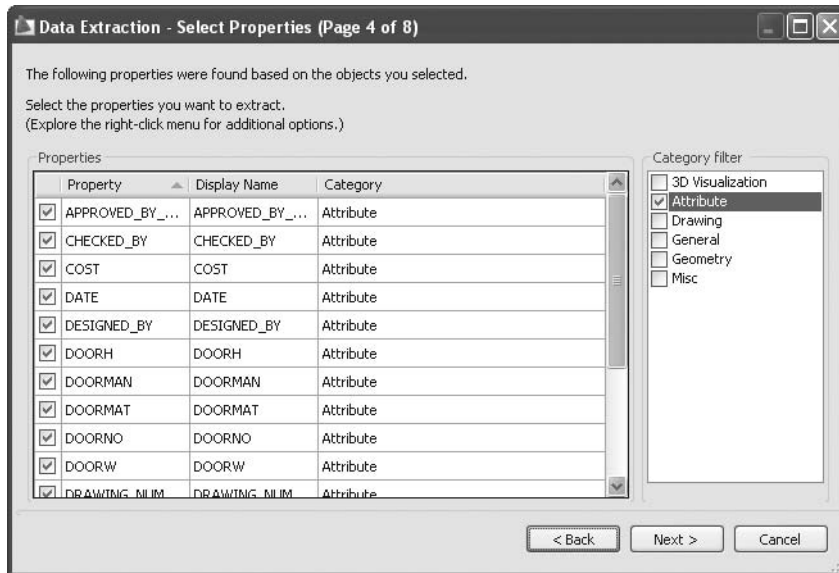
- The following is the third screen:



- Since we are in the process of extracting attributes from blocks, you must make sure that the **Display options** are set as follows:
  - **Display all object types** is off.
  - **Display blocks only** is selected.
  - **Display non-blocks only** is off.
  - **Display blocks with attributes only** is on.
  - **Display objects currently in-use only** is off.
- By default, all blocks are selected. Therefore, if you don't wish to include attributes for certain blocks, this is the right time for such an action.
- Click **Next**.

#### Step Four

- You will see the following screen:

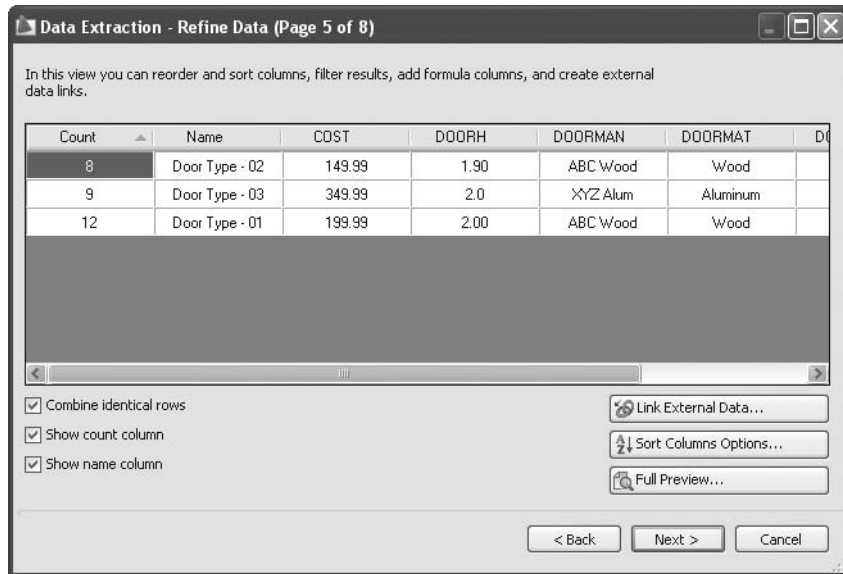


- This is where you will specify the necessary information.
- As you can see from the right portion of this dialog box, there are six categories of information at your disposal. They are:
  - **3D Visualization**
  - **Attribute**
  - **Drawing**
  - **General**
  - **Geometry**
  - **Misc**

- For now we are only interested in extracting the attributes, so we will make sure that all other options are turned off.
- Now you can deselect any undesired attributes.
- Click **Next**.

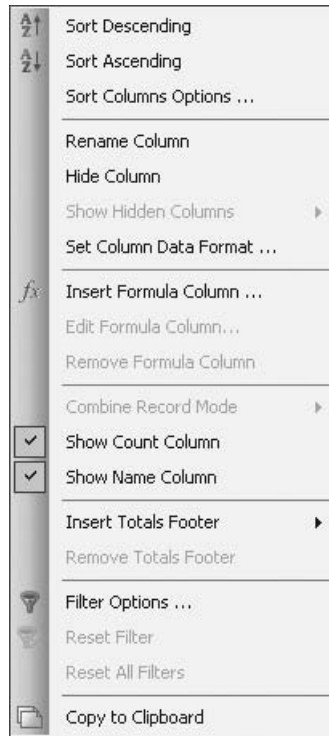
### Step Five

- You will see the following screen:

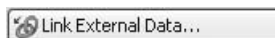


- This is the place to refine the table that you will get.
- Before we discuss the contents of this dialog box, you must understand the following:
  - You can change the width of the columns by dragging the line between any two columns to the left or right.
  - You can change the position of any column by clicking and holding the title of the column, and then dragging it to the new position.
  - You can sort the table according to any column by clicking the title of the column. The first click will be **Ascending**; clicking again will make it **Descending**.

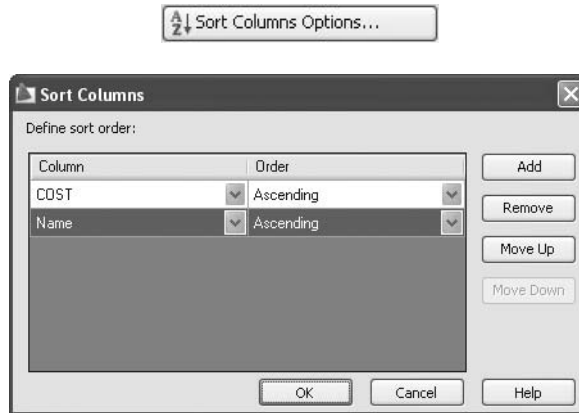
- If you right-click any of the column titles, you will see the following menu:



- Using this menu, you can:
  - Sort columns in either **Descending** or **Ascending** order.
  - **Rename, Hide, or Show** columns.
  - **Set Column Data Format.**
  - **Insert** a new **Formula Column.**
  - Choose to **Show** the **Count** and **Name Columns.**
  - **Insert Totals Footer.**
  - Create a filter and choose **Filter Options.**
- Now establish the following options:
  - **Combine identical rows** allows you to combine the rows that hold the same data.
  - Specify whether or not to show the **Count Column.**
  - Specify whether or not to show the **Name Column.**
- The **Link External Data** option allows you to establish a data link between extracted data and an Excel spreadsheet.



- **Sort Columns Options** is a more sophisticated sorting feature that allows you to assign more than one column to sort according to order. See the following dialog box:



- In the preceding example we assigned two sorting criteria. The first being **COST** in **Ascending** order, and the second being **Name** in **Ascending** order.
- You can add as many criteria as you wish.
- You can move them up or down to indicate priority.
- The **Full Preview** option allows you to see what your table will look like. Click this button and you will see something like the following:

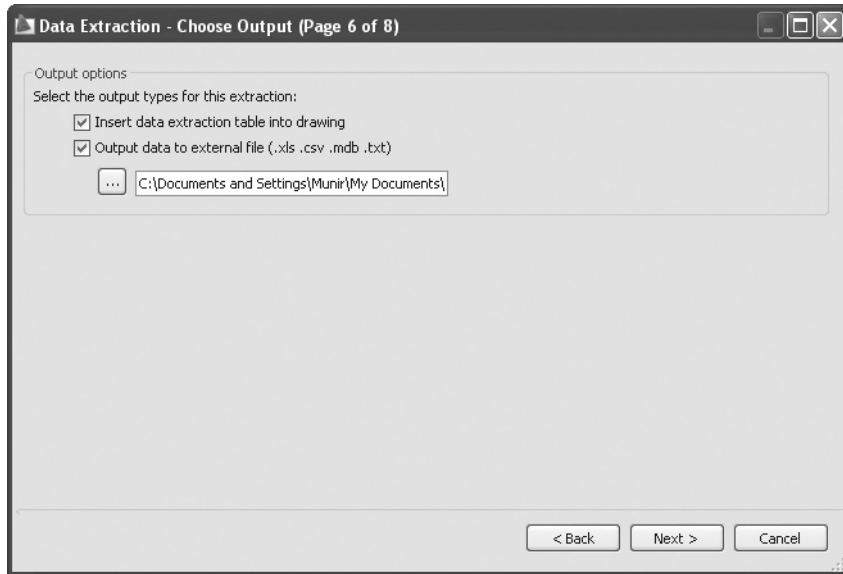


| Count | Name           | COST   | DOORH | DOORMAN  | DOORMAT  | DOORNO | DOORW |
|-------|----------------|--------|-------|----------|----------|--------|-------|
| 8     | Door Type - 02 | 149.99 | 1.90  | ABC Wood | Wood     | 02     | 0.8   |
| 9     | Door Type - 03 | 349.99 | 2.0   | XYZ Alum | Aluminum | 03     | 1.80  |
| 12    | Door Type - 01 | 199.99 | 2.00  | ABC Wood | Wood     | 01     | 0.9   |

- Click **Next**.

**Step Six**

- You will see the following screen:



- In this dialog box, select which type(s) of output you want.
- You can select one or both of the following:
  - **Insert data extraction table into drawing** (it is preferable to create a table style prior to the extraction process).
  - **Output data to external file.** There are four types of file formats (*.xls*, *.csv*, *.mdb*, and *.txt*). If you select this option, choose the desired folder to save the file in.
- Click **Next**.

**Step Seven**

- You will see the following screen:

**Data Extraction - Table Style (Page 7 of 8)**

Table style  
Select the table style to use for the inserted table:  
Door Schedule

Formatting and structure  
☐ Use table in table style for label rows  
☒ Manually setup table  
Enter a title for your table:  
<Title>  
Title cell style: Title  
Header cell style: Header  
Data cell style: Data  
☒ Use property names as additional column headers

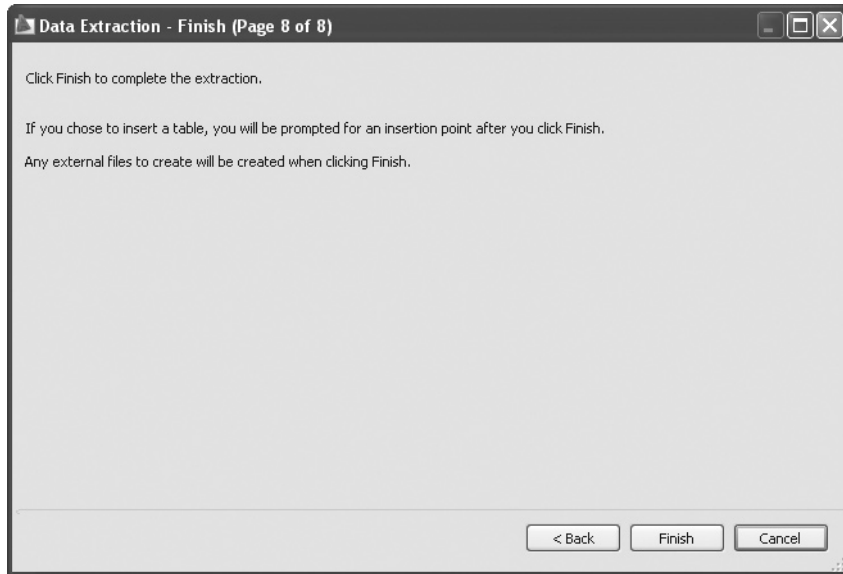
| Title  |        |
|--------|--------|
| Header | Header |
| Data   | Data   |
| Data   | Data   |
| Data   | Data   |
| Data   | Data   |
| Data   | Data   |
| Data   | Data   |

< Back   Next >   Cancel

- If you didn't create the **Table style** prior to this step (recommended), then click the small button and start setting up the table style now.
- Leave the rest as default.
- Click **Next**.

**Step Eight**

- You will see the following screen:



- Click **Finish** to conclude the extraction process.

## EXTRACTING ATTRIBUTES (GENERIC)

---

**Exercise 75**

- Start AutoCAD 2010.
- Open the file *Exercise\_75.dwg*.
- Make sure that the current layer = **Text**.
- Start the extracting process, making the following changes:
  - **Create a new data extraction** and save as **Door Schedule**.
  - Turn **Include current drawing** off.
  - Click the **Add Folder** button and add your exercise folder.
  - Turn **Include Subfolders** off.
  - Turn **Utilize wild-card characters to select drawings** on, and input the following: *\*floor plan.dwg*. Click **OK**. Remove the two files that start with *John Smith*. Only three files should be selected.
  - Turn **Display all object types** off, but make certain **Display blocks only** is on.
  - Turn **Display blocks with Attributes only** on.
  - Turn the block named **ISO A1 title block** off.



- Make sure to select only **Attribute** under **Category filter**. (You should be in Page 5 of 8 right now.)
- Move the **COST** column to the end of the columns.
- Hide the **DOORNO** column.
- Rename the **Count** column to be **Qty** and move it to the left of the **COST** column.
- Rename **DOORH** to be **Door Height** and move it to the right of the **Name** column.
- Rename **DOORW** to be **Door Width** and move it to the right of the **Door Height** column.
- Rename **DOORMAN** to be **Door Manufacturer**.
- Rename **DOORMAT** to be **Door Material**.
- Data format for the **COST** column should be **Currency** with 0.00 precision.
- Right-click the **COST** column and select the **Insert Formula Column**.
- Name the new column **Total Cost**. The formula should be **COST\*Qty** (you must double-click).
- Change the **Data format** to match the **COST** column.
- Sort the table according to **Name** in **descending** order.
- Select **Insert data extraction table into drawing**.
- The title of the table should read **Door Schedule**.
- Insert the table in the **Model space**.
- If you click the table you will notice it is **Locked** and **Linked** (we covered this topic in Chapter 12).
- Save the file and close it.

---

## CHAPTER REVIEW

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1. Which of the following is not a method used to create a dynamic block?
  - a. Double-click an existing block.
  - b. Use the **Block** command.
  - c. Right-click an existing block and select the **Convert to Dynamic** option.
  - d. Use the **Block Editor** command.
2. You need to use the \_\_\_\_\_ command to edit attribute values globally.

3. Which one of these is not an action in **Block Editor**?
  - a. **Stretch**
  - b. **Scale**
  - c. **Fillet**
  - d. **Array**
4. If you redefine a block and add new attributes, you have to use the **Synchronize** command.
  - a. True
  - b. False
5. You can only extract attributes to be a table in a drawing.
  - a. True
  - b. False
6. \_\_\_\_\_ are parameters and actions together.

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## CHAPTER REVIEW ANSWERS

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1. c
2. **Find**
3. c
4. a
5. b
6. **Parameter Sets**

# Chapter 14

## EXTERNAL REFERENCING (XREF)

### In This Chapter

- ◇ What Is External Reference (XREF)?
- ◇ Inserting XREF
- ◇ The Contents of the XREF Palette
- ◇ Layers
- ◇ Edit-in-Place Versus Opening
- ◇ Unload, Reload, Detach, and Bind
- ◇ The **Clipping** Command
- ◇ Clicking on an XREF
- ◇ eTransmit

### **14.1 WHAT IS EXTERNAL REFERENCE (XREF)?**

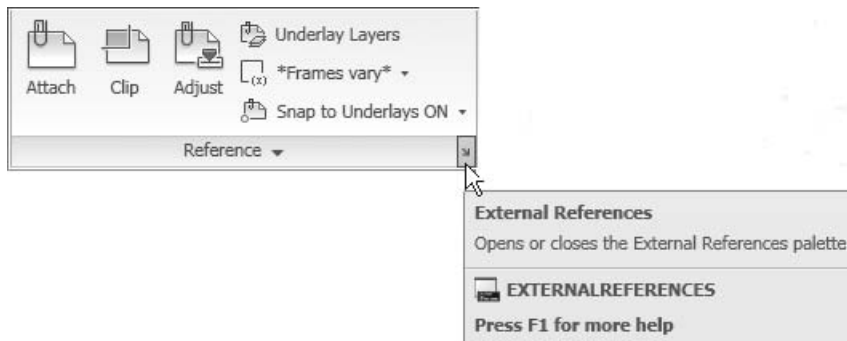
---

- Imagine you are working in a firm and you are doing a layout of beams and columns of the ground floor. You realize you need the drawing that contains the footings of the same building from your colleague. Those drawings are crucial for correct structural layout, and until now the only way you knew how to access those drawings was to use the **Insert** command, which will bring a drawing into another drawing.
- Unfortunately, this method has three problems:
  - You will import the whole file into your drawing; hence, your file size will increase significantly.
  - The incoming file will also import all of its components like **Layers**, **Blocks**, **Text Style**, **Dimension Styles**, and so on, which will make your drawing difficult to control.
  - Additionally, if the owner of the original file made some changes, you will not see them because there is no live connection between the two files.

- **External Reference (XREF)** will allow you to import a *.dwg*, *.dwt*, PDF, raster file, or DGN file to your current file, thereby solving all three problems as follows:
  - Because **XREF** doesn't import the entire file to the current file, but merely a small image of it, the size increase to the current file is incremental.
  - **XREF** separates all of the incoming components from the existing **Layers**, **Blocks**, **Dimension Styles**, and so on. You will see that a layer from an **XREF** file looks something like: **First\_Floor\_Plan|Wall**. The first part is the name of the file, then a separator or |, and then the name of the layer. Using this method, it is easy to distinguish the layers in your file from the layers coming from an **XREF** file.
  - **XREF** maintains a live connection between the original file and its image in your file. Therefore, **XREF** will update the image in your current file if any modification takes place at any time in the original file.

## 14.2 INSERTING XREF

- The first step is to open the desired file that will accommodate the **XREF** file(s).
- To issue the **XREF Insert** command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **References** panel, click the button at the lower right corner of the panel:



- You will see the **External Reference** palette:



- At the top left part of the palette, click the small button and you will see the following menu:

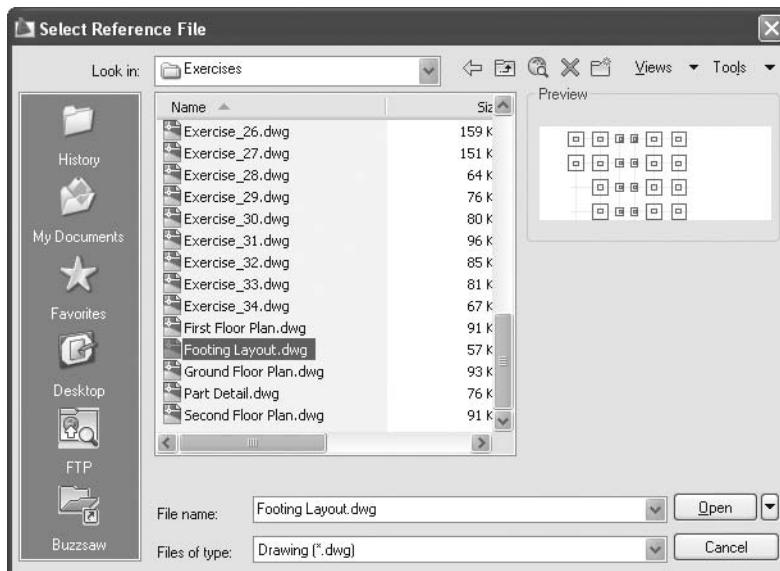


- You can select one of the following file types to import to your current drawing. They are:
  - DWG (any AutoCAD® drawing file)
  - Image (any raster file: \*.jpg, \*.tif, \*.bmp, etc.)

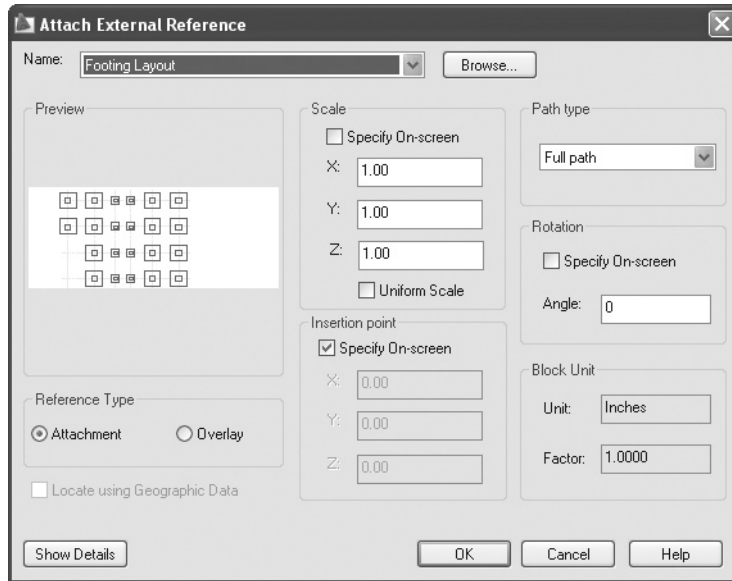
- DWF (any Design Web Format file)
- DGN (any MicroStation® V8 file)
- PDF (any Adobe® Portable Document File)

### Attach DWG

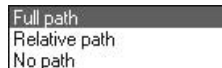
- The **Attach DWG** command will allow you to insert an existing AutoCAD drawing into your current AutoCAD drawing. You will see the following dialog box:



- Select the desired drive, folder, and file, and then click **Open**. The following dialog box will appear:

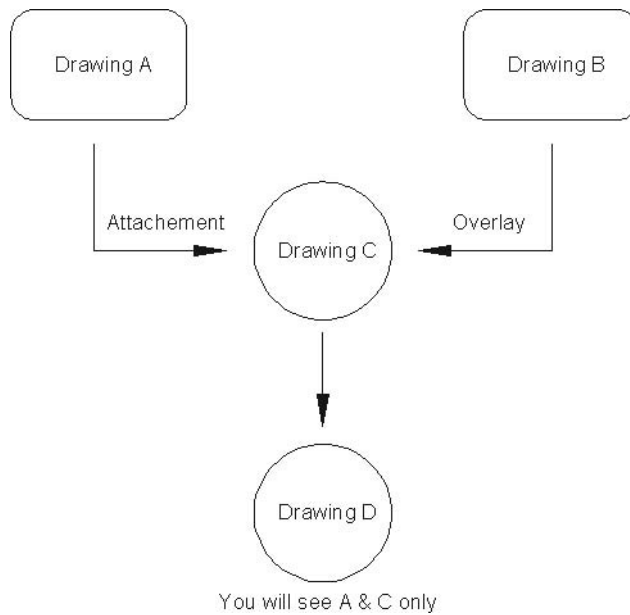


- From the preceding dialog box, you can see that AutoCAD saves the path of the file to track any changes that may take place.
- Specify the **Path type**. You will have three choices:



- The three choices are:
  - **Full path**, which means that AutoCAD will save the exact path of the file. Each time an **XREF** is loaded using this option, AutoCAD will automatically access this specific folder.
  - **Relative path**, which means that AutoCAD will save only the folder that contains the file. Even if you move the folder to another drive, AutoCAD will be able to locate it.
  - **No path**, which means that AutoCAD will not save any path for this **XREF**. Instead, each time an **XREF** is needed, AutoCAD will search the current folder, and then the **Search** paths specified in **Tools/Options** on the **Files** tab (**Project paths** and **Support paths**).

- Specify the **Attachment Type**. Before we proceed, let's look at the following illustration:



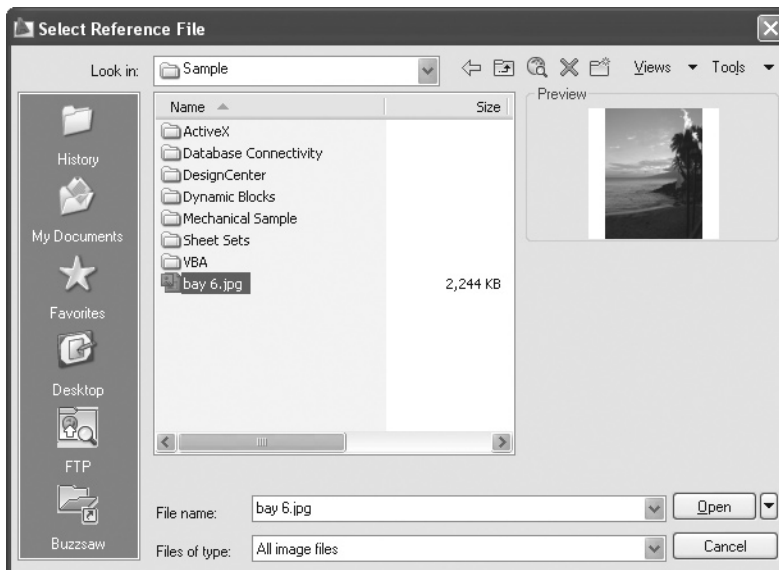
- Based on the preceding illustration, while you are in Drawing C, you imported two drawings: Drawing A and Drawing B. You closed C, opened Drawing D, and then imported Drawing C. What do you see?
- The answer to this question is: you will see Drawing C and Drawing A. You will *not* see Drawing B. Why?
- The reason is **Attachment Type** controls it. Looking at the preceding illustration, Drawing A is an **XREF** as an **Attachment**. Drawing B is an **XREF** as an **Overlay**.
- Drawing B will be shown in Drawing C, but not in any other drawing if Drawing C happened to be imported via **XREF** in another drawing.
- The rest of the dialog box is identical to the **Insert** dialog box.

#### Attach Image

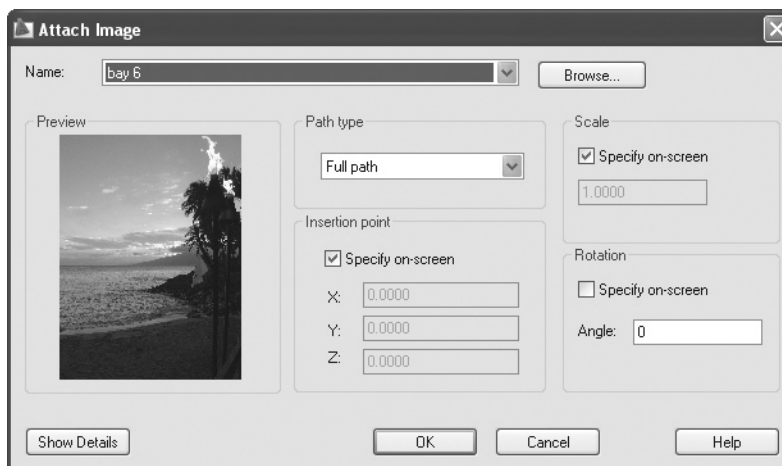
- The **Attach Image** command will allow you to insert a raster file into the current AutoCAD drawing.



- You will see the following dialog box:



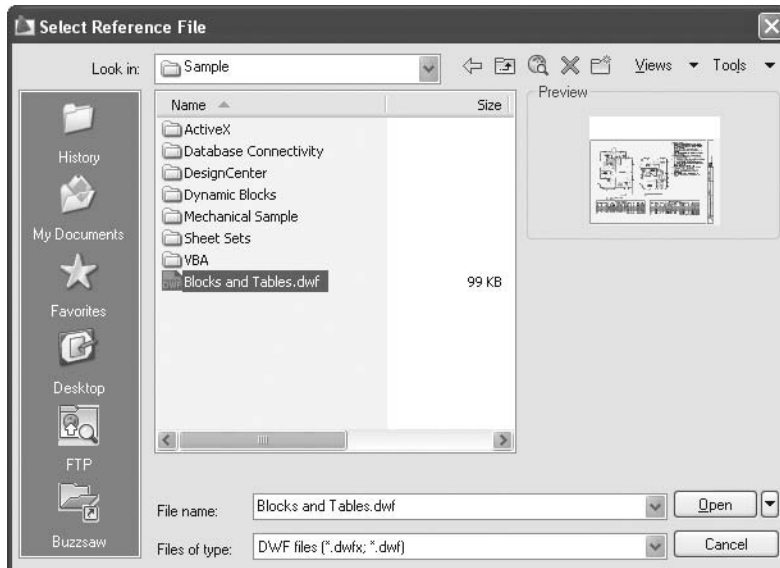
- Select the desired drive, folder, and file, and then click **Open**. The following dialog box will appear:



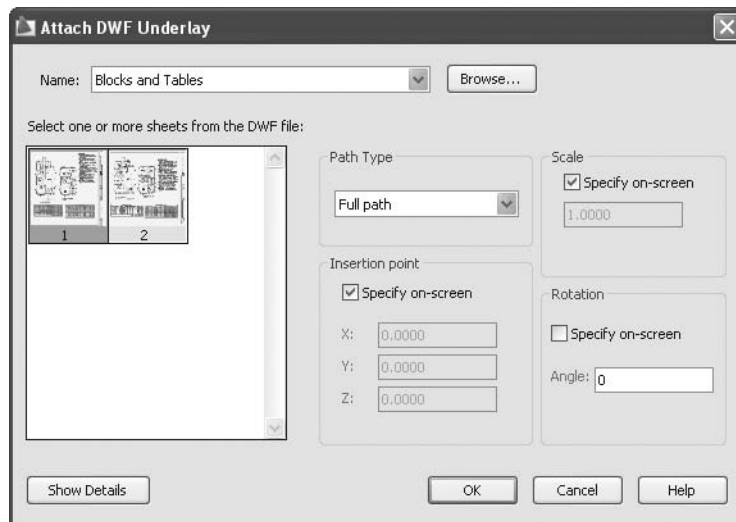
- Each part of this dialog box was discussed in the Attach DWG section.

**Attach DWF**

- The **Attach DWF** command will allow you to insert a Design Web File (DWF) into the current AutoCAD drawing.
- You will see the following dialog box:



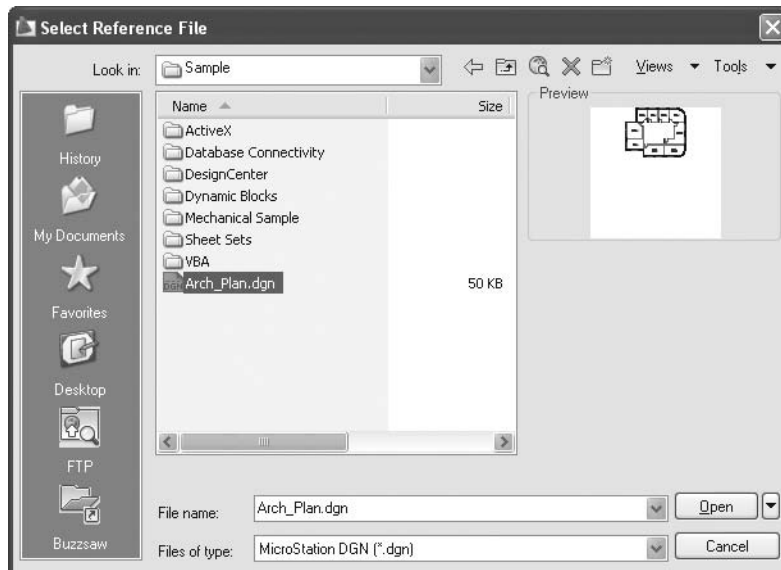
- Select the desired drive, folder, and file, and then click **Open**. The following dialog box will appear:



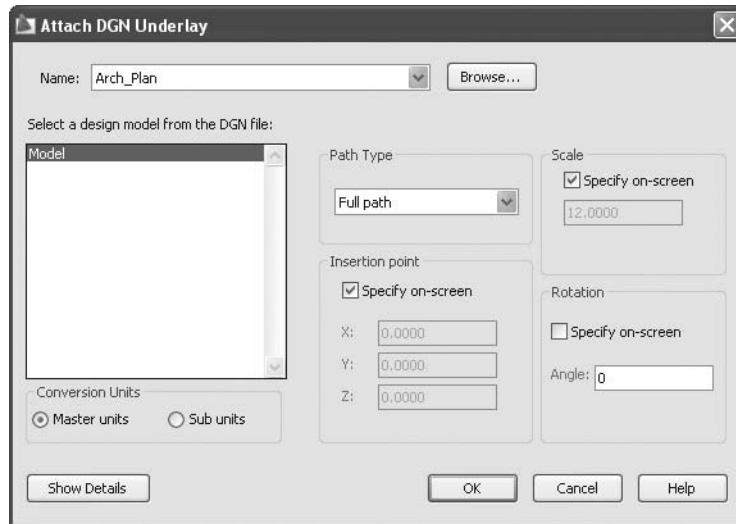
- The interesting thing in this dialog box is the part labeled **Select one or more sheets from the DWF file**. This option allows you to cover all of the possibilities, as there is single-sheet DWF and multi-sheet DWFs. If you have the second type, select one of the sheets to attach to your current file.
- The rest of the dialog box has already been discussed.

### Attach DGN

- The **Attach DGN** command allows you to insert a Microstation file (DGN) into the current AutoCAD drawing.
- You will see the following dialog box:



- Select the desired drive, folder, and file, and then click **Open**. The following dialog box will appear:

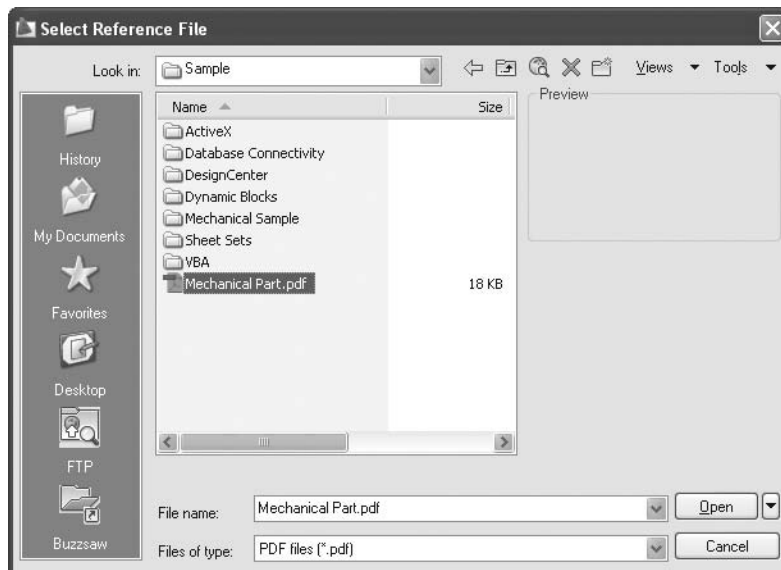


- A Microstation file always contains two units: **Master** and **Sub**. Specify which of the units should be converted to AutoCAD units.
- For example, an AutoCAD unit is designated by either millimeter or inch. Suppose you have a DGN file with **Master units** = feet, and **Sub units** = inch. Which one will you select? The answer is **Sub units**.
- The rest of the dialog box has already been discussed.

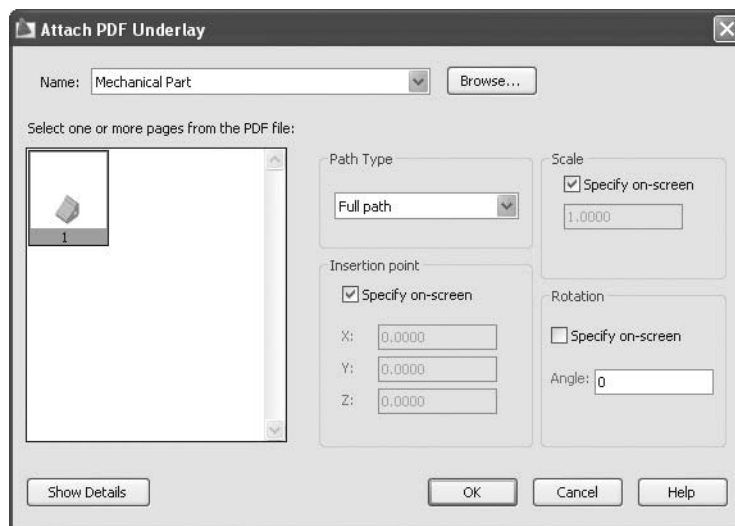
#### Attach PDF

- The **Attach PDF** command allows you to insert a PDF file into the current AutoCAD drawing.

- You will see the following dialog box:



- Select the desired drive, folder, and file, and then click **Open**. The following dialog box will appear:

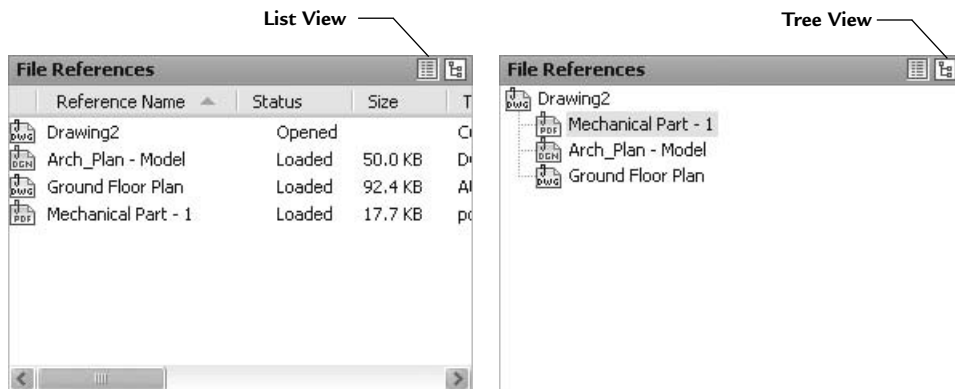




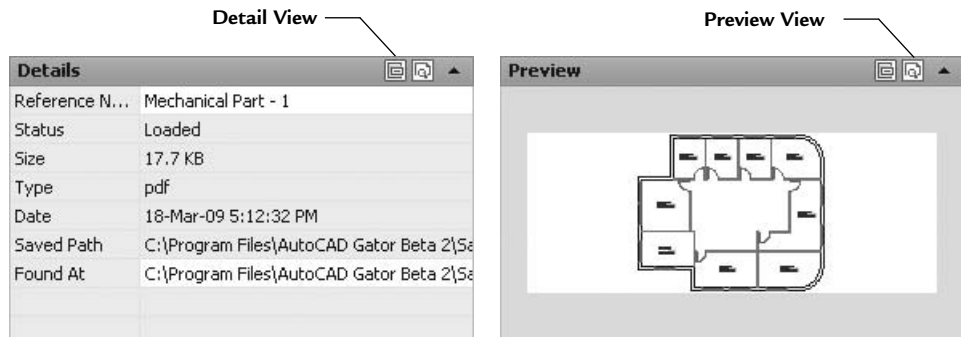
- AutoCAD assumes that the PDF may be multiple pages, so you will be asked to select one or more of these pages.
- The preceding example is for a single-page PDF file.
- The rest of the dialog box has already been discussed.
- You can only attach a single copy of the same DWG file in your current file.
- You can attach multiple copies of the same image, DWF, DGN, and PDF file in your current file.
- The current layer will accommodate the whole **XREF**, so be sure you know which layer was current when you attached the **XREF**.

### 14.3 THE CONTENTS OF THE XREF PALETTE

- There are several parts of the **XREF palette** that you need to understand.
- Each part will help you see pieces of information that differ from one another.
- See the following views:
  - At the top right portion of the palette, you have two small buttons. See the following illustration:



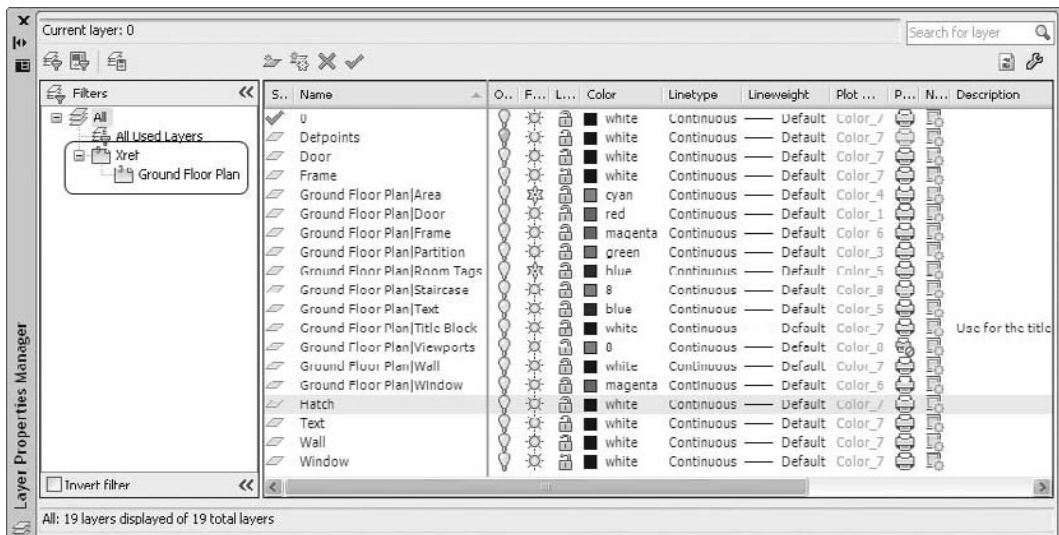
- On the top right side of the palette you have two small buttons. See the following illustration:



- Between the two parts, there is a small arrow at the right. If you click it, you will hide the lower portion, giving more space to the top portion.

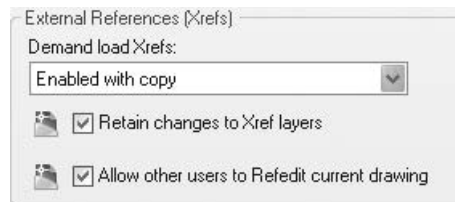
## 14.4 LAYERS

- After you attach an **XREF** file, visit the **Layer Properties Manager**. You will see something like the following dialog box:



- You will see the layers that belong to your file, such as **Frame**, **Door**, etc.

- You will also see **XREF** layers, such as **Ground Floor Plan\Area**, which means you can easily segregate your layers from the layers attached to the **XREF** file. You can also segregate **XREF** layers from one another.
- At the left part of the dialog box, you can see a filter called **Xref**, which is automatically created by AutoCAD.
- Note the following facts about layers and **XREF**:
  - You can't make any of the **XREF** layers current. Therefore, you can't draw on these layers.
  - You can change the color and linetype of an **XREF** layer. Additionally, you can turn it off, freeze it, and lock it. By default, all of these changes will be saved. If you do not want the changes to be saved, use the system variable **VISRETAIN**. By default, **VISRETAIN** = 1, so set the value to 0 (zero). Another method is through the **Options** dialog box. Use the **Open and Save** tab, and turn off the option **Retain changes to Xref layers**. See the following illustration:



## ATTACHING XREF (GENERIC)



### Exercise 76

- Start AutoCAD 2010.
- Open the file *Exercise\_76.dwg*.
- Using the **XREF palette**, attach the following:
  - Attach *Footing Layout.dwg*, using 0,0. Leave the other parameters as default.
  - Attach *Architectural Layout.dwf*, using 0,0. Leave the other parameters as default.
  - In the **Architectural** layout, attach the image *Rendered Presentation.jpg* in the square at the upper right corner of the layout, using **Specify on-screen** for both the **Insertion point** and **Scale**.
- Go to the **Layer Properties** palette and see the layers.
  - Can you differentiate between the layers in the drawing and the layers coming from the **XREF** file?
  - Does the DWF file or the image file import any layers with them?



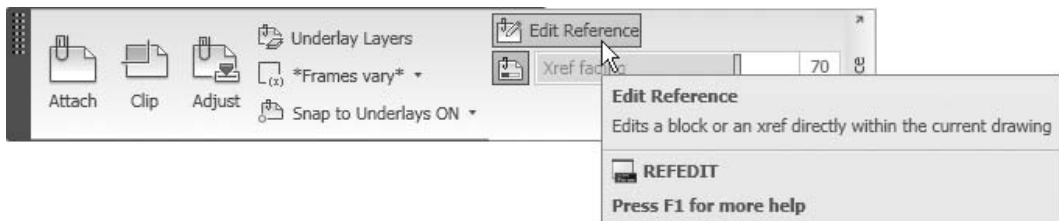
- Change the color of the layer **Footings Layout** from red to magenta (if needed, use the **REGEN** command).
- Go to **Options** and make sure **Retain changes to XREF layer** is turned off.
- Save the file and close it.
- Reopen the file. What is the color of **Footings Layout** and why?
- Make layer **Columns** the current layer, then freeze layer 0. What will happen and why?
- Thaw layer 0.
- Save the file and close it.

## 14.5 EDIT-IN-PLACE VERSUS OPENING

- Assume you attached a DWG file and then you found that your colleague made a mistake in his or her file.
- AutoCAD gives you the ability to change the attached file without leaving your file (i.e., **Edit-In-Place**). Or, you can open the file and edit it (i.e., **Open**).

### Edit-in-Place

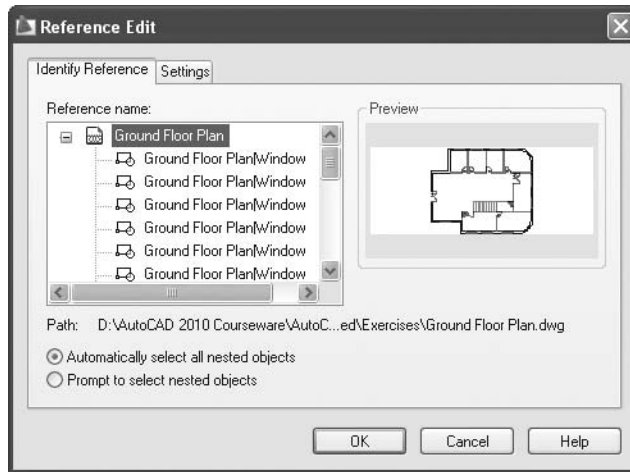
- To issue this command, make sure you are on the **Insert** tab on the **Ribbon** and, using the **References** panel, click the **Edit Reference** button:



- AutoCAD will prompt you to:

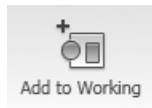
Select reference:

- Select the **XREF** file. The following dialog box will appear:



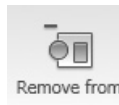
- Click **OK**. Once you do that, all of the drawing will be dimmed (you can't edit the dimmed objects) except for your **XREF** objects.
- A new panel will appear called **Reference Edit**. It has four buttons. The following is a description of each.
- Click the **Add to Working** button. The following prompt will appear:

Select objects:

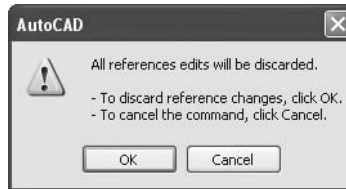


- Select the desired objects to be added.
- Click the **Remove from** button. The following prompt will appear:

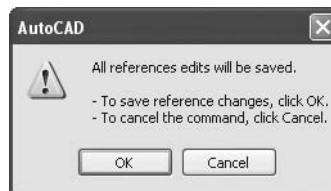
Select objects:



- Select the desired objects to be removed.
- When you are done with your editing, you can either discard the changes as if nothing happened, or save the changes to the original block. Use either **Discard** or **Save** (see the following discussion).
- Click the **Discard** button. The following dialog box will appear:

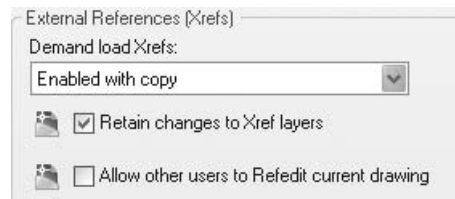


- If you really wish to discard all the changes you made to the block, simply click **OK**. Otherwise, click **Cancel**.
- Click the **Save** button. The following dialog box will appear:



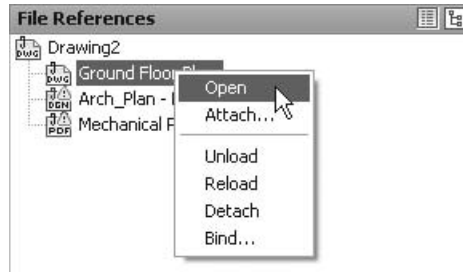
- Click **OK** if you are sure of the changes you made. Otherwise, click **Cancel**.
- Because the previous step is dangerous to do with any file that belongs to a colleague, they will have the power to prevent you or any other user from editing his or her file.
- To do this, click the **Options** button and select the **Open and Save** tab in the **External reference (Xrefs)** panel. Turn off the option **Allow other users to Refedit current drawing**, as in the following illustration:

NOTE

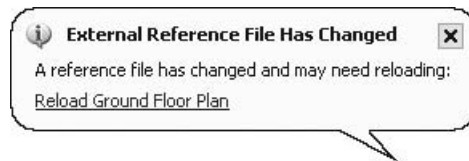


### Opening XREF

- Another method is to open the **XREF** file, edit it, and then save and close it.
- Select the desired file in the **XREF palette** and right-click it. From the file menu, select **Open**. See the following illustration:



- You will see the open **XREF** file and you can edit it as you wish. Once you are done, save the file and close it.
- An alert will inform you that some changes took place in one of the files you are importing via **XREF**. You may need to reload (we will discuss this topic shortly).

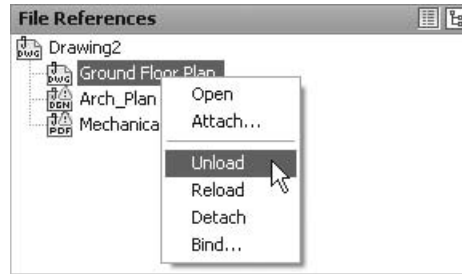


## 14.6 UNLOAD, RELOAD, DETACH, AND BIND

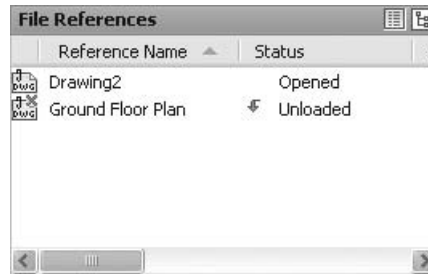
- While you are in the **XREF palette**, you can select any **XREF** file and right-click. Accordingly, you can do one or all of the following:
  - **Unload**
  - **Reload**
  - **Detach**
  - **Bind**

### Unload

- The **Unload** command allows you to unload the **XREF** file, while keeping the link.
  - Using the **External Reference** palette, select the desired **XREF** file, right-click, and then select **Unload**.



- You will see something like the following:



### Reload

- You can use the **Reload** command to do multiple things.
  - It will allow you to reload an unloaded **XREF** file.
  - It will allow you to reload a file that you opened, edited, and saved.
- To issue this command, select the desired **XREF** file, right-click, and then select **Reload**.

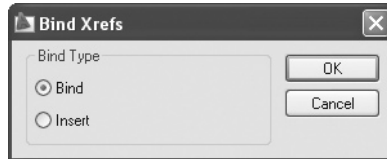
### Detach

- The **Detach** command is identical to the **Unload** command, except that the link is lost in the process.
- When using the **Detach** command, you must attach **XREF** again, unlike the **Unload** command, which you can reload.

### Bind

- Binding will allow you to put physical objects like layers, styles, and blocks in the current file, and will cut any link to the original file.
- There are two binding types:
  - **Bind**
  - **Insert**
- Both types will function in the same way except for the naming of the layers, styles, and blocks.

- Assume we have a layer with the name **Basement Plan|Deck**. The two types will change the name as follows:
  - If you use **Bind**, the name will become **Basement Plan\$0\$Deck**.
  - If you use **Insert**, the name will become **Deck**.
- To use the **Bind** command, select the desired **XREF** file from the **XREF** palette, right-click, and then select **Bind**. You will see the following dialog box:



- Select the desired type and click **OK**.

## EDITING XREF (GENERIC)

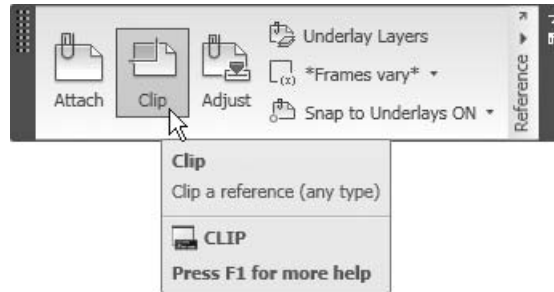


### Exercise 77

- Start AutoCAD 2010.
- Open the file *Exercise\_77.dwg*.
- Make layer **COL-LAYOUT** the current layer.
- Attach the file *Column Layout.dwg* using point 0,0, leaving the rest as defaults.
- As you can see, all the columns fit except the one at the lower right; it is drifted to the left.
- Click one of the columns. When the **External Reference** tab appears, click **Edit Reference In-Place**. When the dialog box appears, click **OK**.
- Zoom to the column you should move, and using the **Move** command, move it to the right at **distance** = 0.15, and then zoom out.
- Click the **Save** button on the **Edit Reference** panel. Click **OK** in the dialog box.
- Save the file.
- Using the **External Reference palette**, select the **XREF** column, right-click, and then select **Unload**. What happened?
- Select it again and right-click, then select **Reload**. What happened?
- Select the Column layout **XREF**, right-click, and then select **Bind**. In the dialog box, select the **Bind** option. You should have two layers similar to each other, write their names:
  - \_\_\_\_\_
  - \_\_\_\_\_
- Save the file and close it.

## 14.7 THE CLIPPING COMMAND

- When you import an **XREF** file, all of the file will be shown.
- You can elect to show only part of the file by clipping it.
- There is only one **Clip** command for all types of **XREF** files. Once you select the file, AutoCAD will display the appropriate options. Make sure you are on the **Insert** tab on the **Ribbon** and, using the **Reference** panel, click the **Clip** button.



- Another way to issue the same command is to select the desired **XREF** and then right-click. The proper command will be displayed for each type.
- We will discuss the clipping of a DWG as an example. The same method is used for all types of files.
- Once the command is issued, the following prompt will appear:

```
[ON/OFF/Clipdepth/Delete/generate Polyline/New boundary]
<New>:
```

- By default, the option selected is **New boundary**. This option will create a new clipping boundary for the selected **XREF** file. Press [Enter] to accept the default option and the following prompt will appear:

```
[Select polyline/Polygonal/Rectangular/Invert clip]
<Rectangular>:
```

- Choose one of the following methods to access the clipping boundary.
  - Choose the default option **Rectangular** to draw a rectangle around the desired area, and the rest of the file will disappear.
  - Choose the **Select polyline** option, and if you have already drawn a polyline before issuing the **Clip** command, just select it.
  - Choose the **Polygon** option to draw an irregular polygon around the desired area, and the rest of the file will disappear.
- After specifying the boundary, the **Clip** command will end automatically, showing only the area you specified.

- The other options available are **Off**, **On**, **Clipdepth**, **Delete**, **Generate Polyline**, and **Invert Clip**.

#### Off

- The **Off** option allows you to turn off the clipping boundary, thereby showing the complete **XREF** file.

#### On

- The **On** option cancels the off function.

#### Clipdepth

- The **Clipdepth** option will allow you to control the front and back clipping planes in the Z-direction. Since we are dealing with 2D only (i.e., an XY drawing) this option is not applicable.

#### Delete

- The **Delete** option allows you to delete the clipping boundary. (Using the **Erase** command will not remove the clipping boundary.)

#### Generate Polyline

- The **Generate Polyline** option is helpful if the **XREF** frame is turned off. You can draw a polyline identifying the clipping boundary.

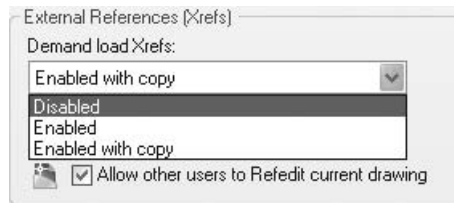
#### Invert Clip

- The **Invert Clip** option is the opposite of the clipping boundary. Instead of showing only the specified area and hiding the rest, by utilizing this option, the area you specify will be hidden and the rest will be shown.

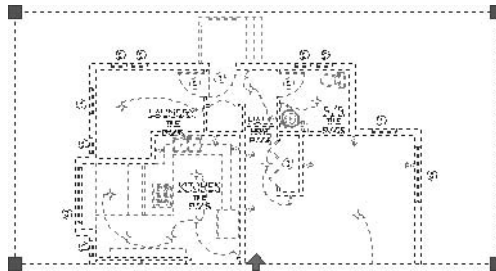
#### Note

- While clipping is a feature used to show some of the **XREF** file and hide some of it, it will not affect the performance of the AutoCAD file by reducing the **XREF** file size in your current file. You have to turn on the **Demand Loading** option in order to benefit from clipping.
- From the menu browser, select the **Options** button and then select the **Open and Save** tab. In **External References (Xrefs)** change the settings of **Demand Load Xrefs**. See the following illustration:

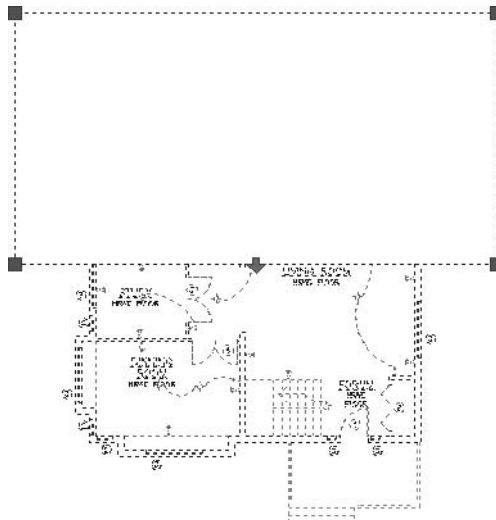




- You are presented with three options to choose from:
  - **Disabled**, which turns off this feature.
  - **Enabled**, which enables this feature but will not allow users to edit the file.
  - **Enabled with copy**, which enables this feature but allows users to edit the file.
- You can flip between what is hidden and what is shown using an arrow. See the following illustrations:
  - The original drawing:

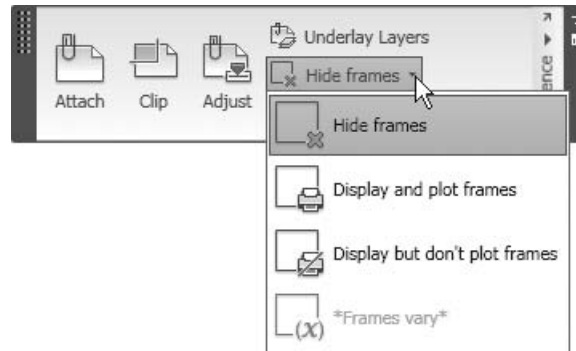


- The drawing after clicking the arrow at the bottom:





- In the **Reference** panel, you can show or hide the clipping frame by using the **Frames** button:



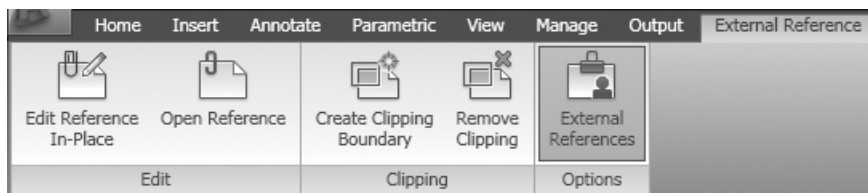
- You can **Hide frames**, **Display and plot frames**, or **Display but don't plot frames**.

## 14.8 CLICKING ON AN XREF

- If you click on an **XREF** (regardless of its type) AutoCAD will display a new tab with panels to edit and control this **XREF**.
- For each **XREF** type, there will be a different tab and panels.
- We will go through each one of the five types in the following discussion.

### DWG XREF

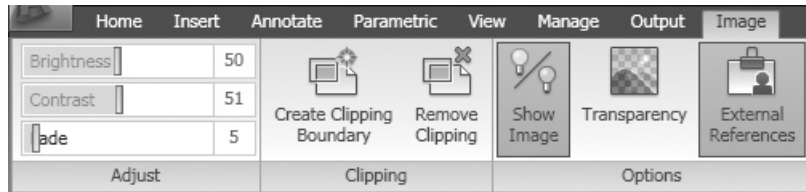
- If you click a **DWG XREF**, you will see the following tab and panels:



- The buttons available are:
  - **Edit Reference In-Place**
  - **Open Reference**
  - **Create Clipping Boundary**
  - **Remove Clipping**
  - **External References** (to show the **External Reference palette**)
- We have covered all of these commands in previous sections.

### Image XREF

- If you click an **Image XREF**, you will see the following tab and panels:



- The **Adjust** panel includes three buttons: **Brightness**, **Contrast**, and **Fade**, all of which control the image sharpness and clarity.
- The **Clipping** panel has already been discussed.
- The **Options** panel includes:
  - **Show Image**, which can show or hide the image.
  - **Transparency**, which can change the background (if it exists) from black to transparent.
  - **External References**, which can show the **External Reference palette**.

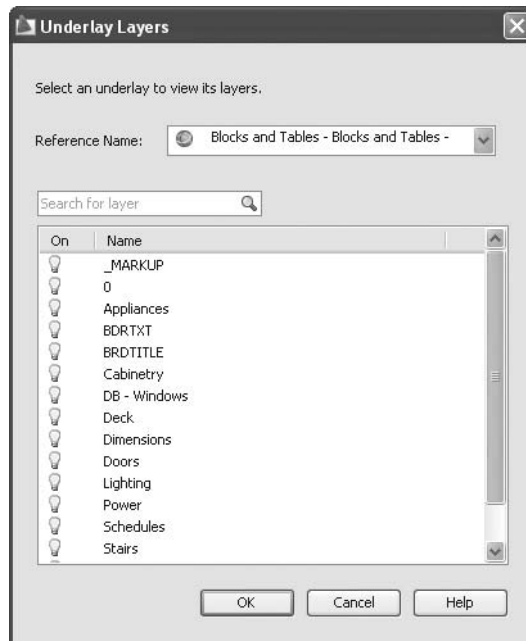
### DWF XREF

- If you click a **DWF XREF**, you will see the following tab and panels:



- The **Adjust** panel includes the **Contrast** and **Fade** buttons, which control the image sharpness and clarity. Also, you can choose between displaying the DWF in color or monochrome.
- The **Clipping** panel has already been discussed.
- The **Options** panel includes:
  - **Show Underlay**, which can show or hide the DWF.
  - **Enable Snap**, which can enable or disable the object snap to the DWF objects.
  - **External References**, which can show the **External Reference palette**.

- The **DWF Layers** panel includes the **Edit Layers** button (the creator of the DWF should include layers with the file). You will see the following dialog box:



### DGN XREF

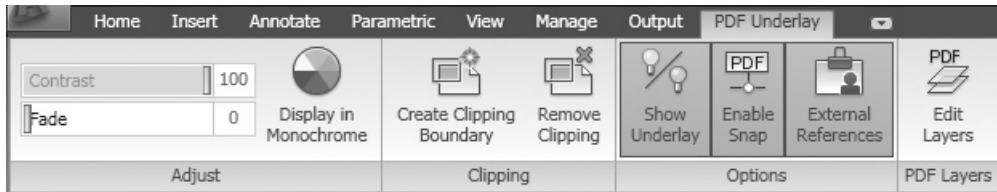
- If you click a **DGN XREF**, you will see the following tab and panels:



- These options are identical to those presented when clicking on a **DWF XREF**.

**PDF XREF**

- If you click a **PDF XREF**, you will see the following tab and panels:



- These options are identical to those presented when clicking on a **DWF XREF**.
- NOTE ■ These functions are the same functions mentioned in the discussion of the original **XREF** panel.

## CLIPPING AND CLICKING (GENERIC)

**Exercise 78**

- Start AutoCAD 2010.
- Open the file *Exercise\_78.dwg*.
- Make sure you are on the **Insert** tab on the **Ribbon** and, using the **Reference** panel, click the **Clip** button. Select one of the footings and create a new rectangular boundary for the four footings on the top right.
- Select one of the four footings and a new panel will appear. Select **Remove Clipping**, and then press [Esc].
- Select one of the green lines (these are DWF objects) and a new panel will appear. Click the **Enable Snap** button. Measure the length of the inner vertical line at the right, what is the distance? (Answer: 14.6973)
- Go to the **Architectural** layout.
- Zoom to the upper right-hand corner of the layout.
- Select the image (if for any reason you can't, press [Ctrl] + A, and then deselect the frame).
- Set **Brightness** = 75, **Contrast** = 27, and **Fade** = 20.
- Clip the image to remove some of the background.
- Save the file and close it.

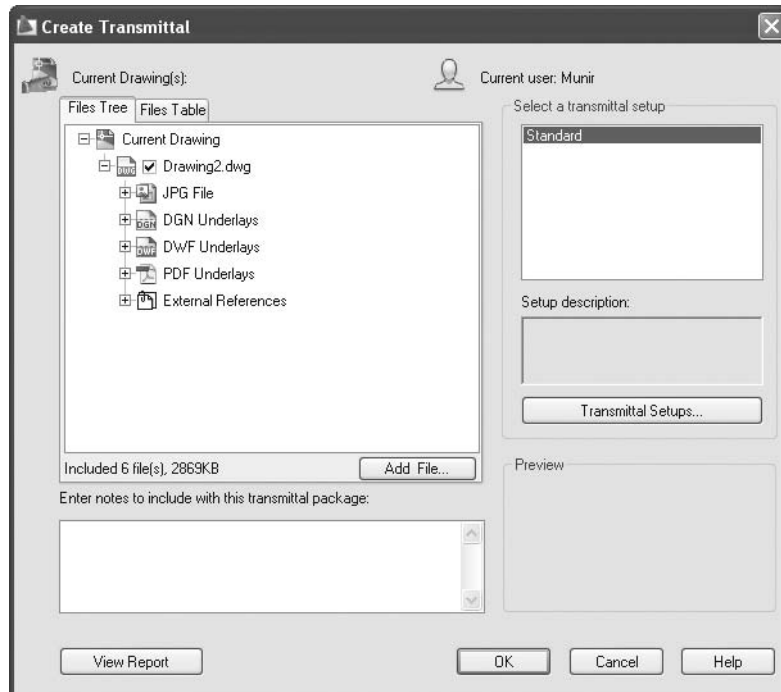
---

## 14.9 eTRANSMIT

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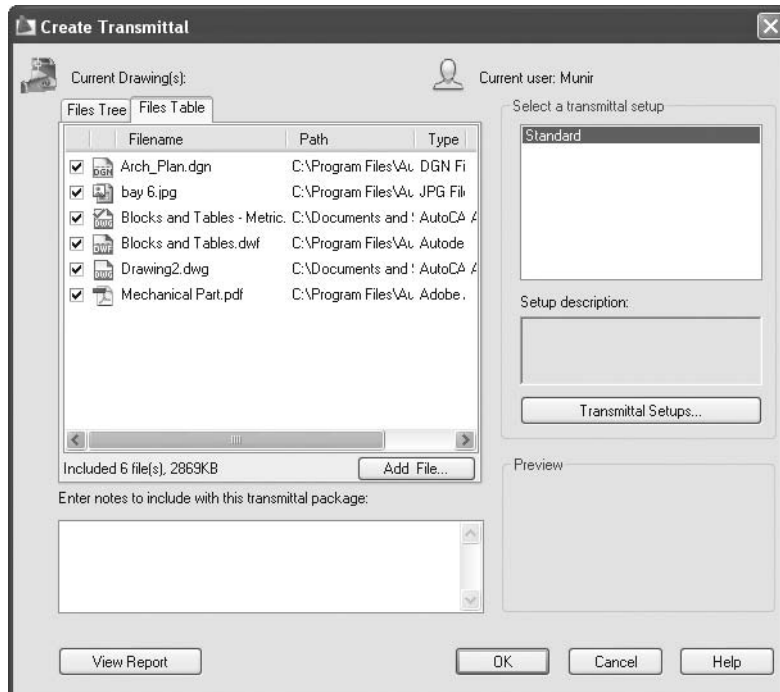
- Sending AutoCAD files from one company to another, whether local or international, is routine for companies all over the world.
- Since **XREF** for these companies means collaboration, they will be using it heavily in their daily work.
- If you want to send a file that has several **XREF**s attached to it, and each **XREF** file may already contain other **XREF** files, can you imagine how much work would be involved in sending this file?
- You would need to search for all of these files and their nested files (if they exist) manually, and group them together. You would then need to use a utility to compress them and create a \*.zip file or an \*.exe file for the receiver that doesn't have WinZip or WinRAR to uncompress the files.
- Suppose you have 100 files you need to send within half an hour. What would you do?
- AutoCAD comes with a convenient solution called **eTransmit**, which solves all of the preceding issues.
- Through eTransmit you can:
  - Select a file to send and AutoCAD will automatically collect all the attached files, regardless of where they reside.
  - eTransmit will assemble the font, hatch, linetype, and other files so the receiver of the file(s) won't need any external help to open your file.
  - eTransmit will compress all of these files (\*.zip or \*.exe) and make them ready for you to send them via any of the communication tools you have.
- Using the **Application menu**, select **Send/eTransmit**.

- You will see the following dialog box:

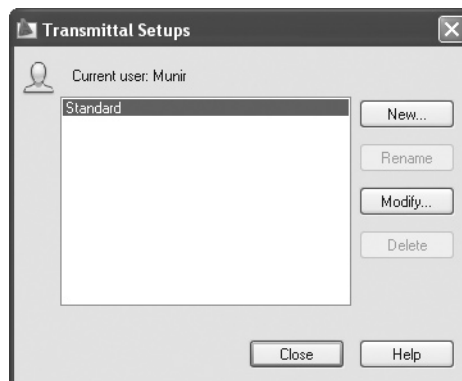


- You can see from the **Files Tree** tab that all the files attached to your current file have been gathered.
- In our example, the **Files Tree** contains a **Font** file, a **JPEG Image**, a **DGN Underlay**, a **DWF Underlay** (another name for **DWF XREF**), a **PDF Underlay**, and a **DWG file**.

- Another view is the **File Table** tab:

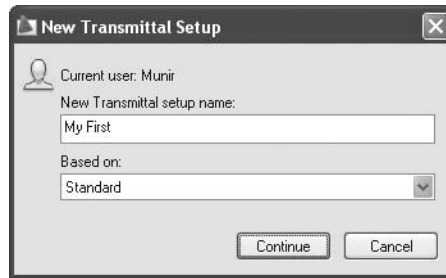


- If you want to include more files in the transmittal package, you will need to click the button **Add File**. A normal file dialog box will be opened to select the desired file(s).
- In order to specify how to create the transmittal package, click the **Transmittal Setups** button. The following dialog box will appear:

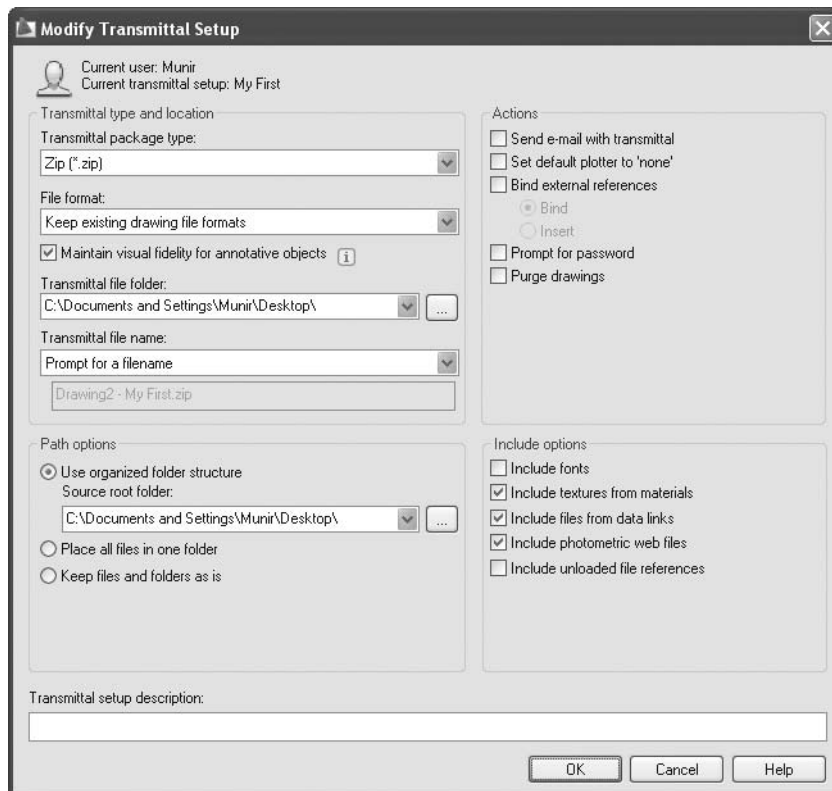




- Click the **New** button to name and create a **New Transmittal Setup** (which can then be used every time):



- Click **Continue**. You will see the following dialog box:



- Select **Transmittal package type** from \*.zip, \*.exe, or a folder containing the file.
- The rest of the options are self-explanatory.
- Once you are done, click **OK**, and a file will be created. You can send it to anyone without the fear that the file has some missing data attached to it.

- At the end it is suggested to look at the report that AutoCAD will produce for you. This report will sum up all the information needed for the transmittal package. You will see something like the following:



## eTRANSMIT (GENERIC)



### Exercise 79

- Start AutoCAD 2010.
- Open the file *Exercise\_79.dwg*.
- Start the **eTransmit** command.
- Make sure that all of the **XREF** files are there.
- Check both the **Files Tree** and the **Files Table**.
- Create a new transmittal setup (call it **My First**) that includes:
  - **Transmittal package type** = Self-extracting executable (\*.exe)
  - **File format** = AutoCAD 2007
  - **Include fonts** = on
- Create the transmittal package.
- What is the size of the \*.exe file? (Answer: almost 600 kb)
- Save the file and close it.

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**CHAPTER REVIEW**

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1. Which one of these files cannot be inserted inside a drawing file by **XREF**?
  - a. DWG
  - b. PDF
  - c. XLS
  - d. DGN
2. You can use the \_\_\_\_\_ command to see some of the **XREF** files.
3. You can control the DWF layers if they were imported into the file via **XREF**.
  - a. True
  - b. False
4. Which one of the following statements is false?
  - a. You can edit any DWG file imported via **XREF** using **Edit-in-Place**.
  - b. You can open any DWG file imported via **XREF**.
  - c. You can change DWF and PDF files if they were imported into your drawing via **XREF**.
  - d. You can bind any DWG file imported via **XREF** to your current drawing.
5. **Unload** and **Detach** are equal commands.
  - a. True
  - b. False
6. The best tool to send any file that uses **XREF** files is \_\_\_\_\_.

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**CHAPTER REVIEW ANSWERS**

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1. c
2. **Clipping**
3. a
4. c
5. b
6. eTransmit



# Chapter 15 SHEET SETS

## In This Chapter

- ◇ Sheet Sets: An Introduction
- ◇ Opening and Closing Existing Sheet Sets
- ◇ Understanding the **Sheet Set Manager** Palette
- ◇ Sheet Sets and Files
- ◇ Creating a New Sheet Set Using an Example
- ◇ Adding Views
- ◇ Creating a New Sheet Set Using Existing Drawings
- ◇ eTransmit, Archive, and Publish Sheet Sets
- ◇ Label Blocks and Callout Blocks

## 15.1 SHEET SETS: AN INTRODUCTION

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- When an Architect Engineer Consultant (AEC) company designs a building, the most important output would be the set of drawings that includes elements such as Architectural, Structural, Mechanical, Electrical, and so on.
- According to the U.S. National CAD Standard (U.S. NCS) there should be an order for the sheets, beginning with General, and following with Hazardous Materials, Surveying/Mapping, Geotechnical, Civil Works, and so forth.
- Also, U.S. NCS requires Sheet Identification (which is also known as Sheet numbers). Complicated projects can use Sheet Identification with multiple parts, and small projects can use abbreviated Sheet Identification (to find out more about NCS you can visit their website at <http://www.buildingsmartalliance.org/ncs/>).
- Setting all of these requirements manually would prove to be tiresome.
- AutoCAD® provides a tool that allows you to create a sheet set that adapts U.S. NCS (or any other system) smoothly and easily.
- This tool is also helpful for publishing (printing) and archiving your projects.

- AutoCAD can help create the sheet sets in two ways:
  - **Example sheet set** uses a common organizational structure of sheets for different disciplines. AutoCAD will create sheet sets without sheets. You create the sheets and then add stored views into these sheets as **XREFs**, which are then scaled to the desired scale factor.
  - **Existing drawings** include already-created layout(s) in each drawing that contain viewports, company frame borders, and all other details. This method considers each layout as a sheet. This is easier, faster, and more time efficient.

## 15.2 OPENING AND CLOSING EXISTING SHEET SETS

- There are two ways to open a sheet set:
  - While no file is open
  - While at least one file is open

### No File Is Open

- If there is no file open, take a look at the **Quick Access** toolbar. In the upper left-hand corner of the screen, you will see the following:

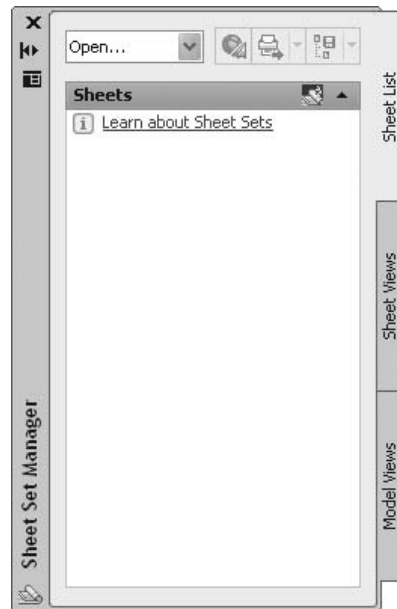


### One File Is Open

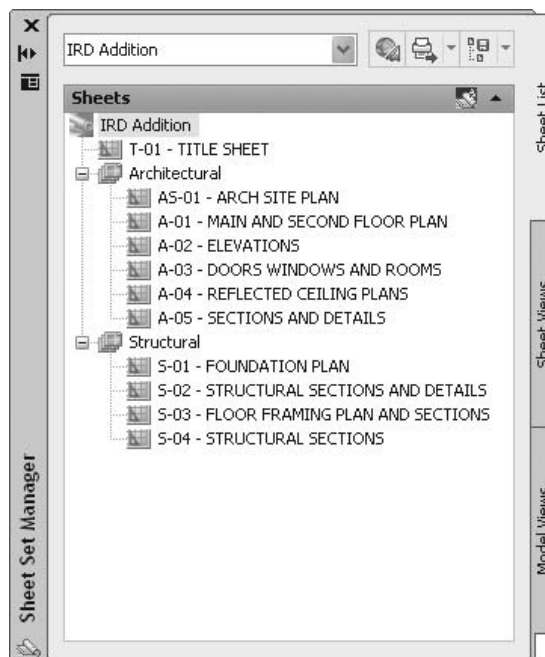
- Make sure you are on the **View** tab on the **Ribbon** and, in the **Palettes** panel, click the **Sheet Set Manager** button.



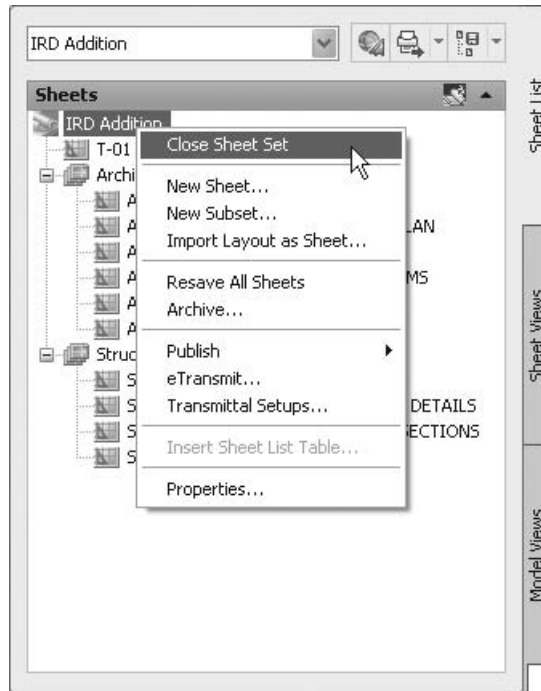
- In both methods, you will see the **Sheet Set Manager** palette:



- From the upper pop-up menu, select **Open**. A normal dialog box will be displayed. Browse to the desired location in your computer (or network) and then select a file to be opened. The file extension for the sheet set is \*.DST.
- You will see something like the following:



- To close the sheet set, simply select its name, right-click, and then select the **Close Sheet Set** option.



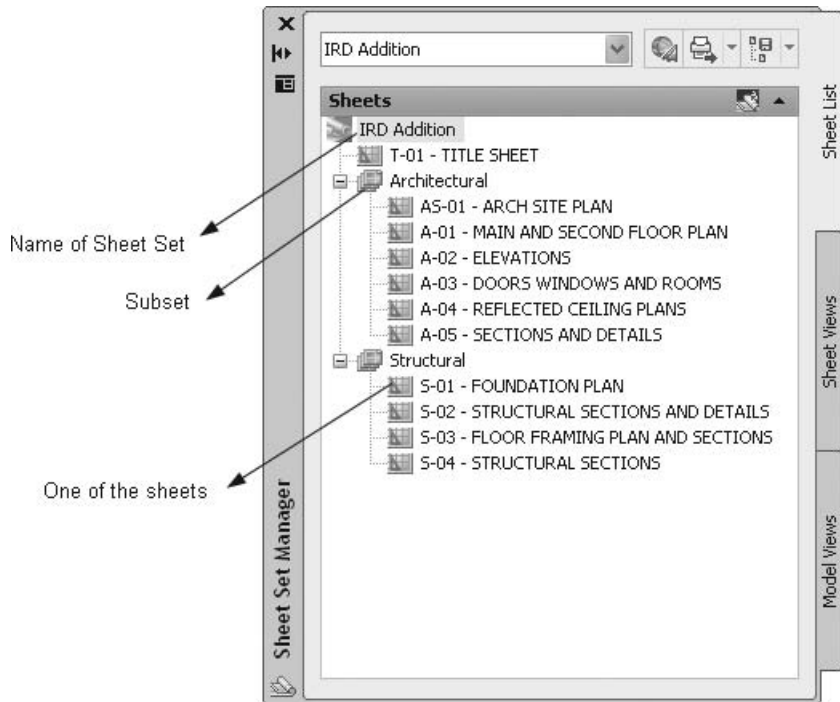
### **15.3 UNDERSTANDING THE SHEET SET MANAGER PALETTE**

- There are three tabs in the **Sheet Set Manager** palette:
  - **Sheet List**
  - **Sheet Views**
  - **Model Views**



### Sheet List

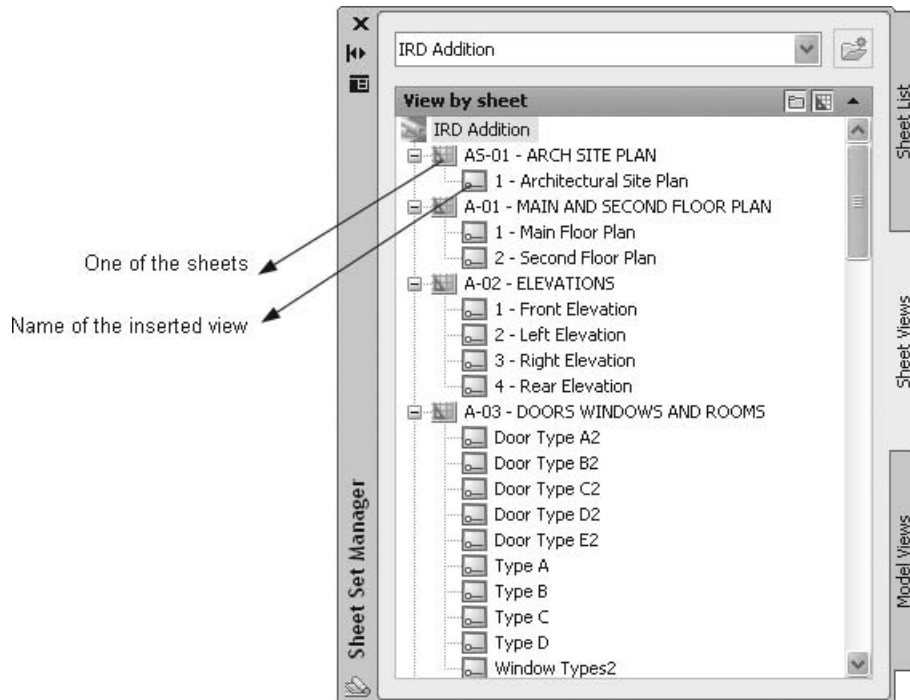
- You will see the following:



- In the **Sheet List** tab, you will see the sheets in the desired print order.
- The hierarchy is as follows:
  - Sheet set name
  - Subsets
  - Sheets (may be linked to the sheet set directly or listed under a subset)
- Subsets are a way of arranging your sheets in groups. This arrangement is optional, but highly recommended.
- Each sheet has a number and a name. See the following example:
  - AS-01 ARCH SITE PLAN

### Sheet Views

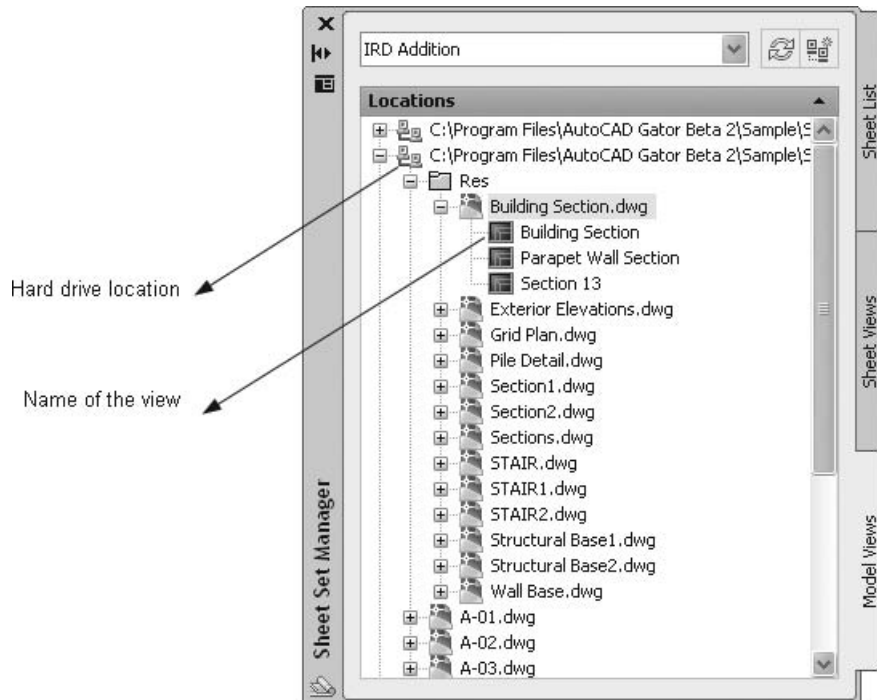
- You will see something like the following:



- In this tab, you will see the list of views inserted (by the creator of this sheet set) in each sheet.

### Model Views

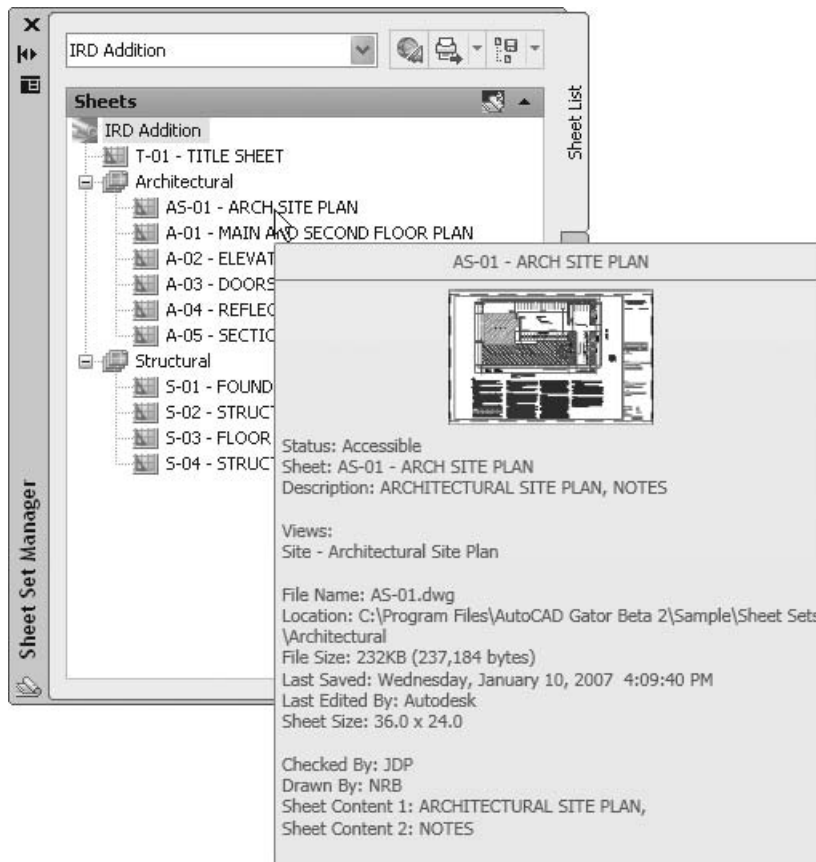
- You will see something like the following:



- In the **Model Views** tab, you will see the hard drive and the folder(s) that contain all of your drawings. You can also see the drawings and any stored views.
- In this tab, you can drag and drop views to an empty sheet (we will discuss this later).

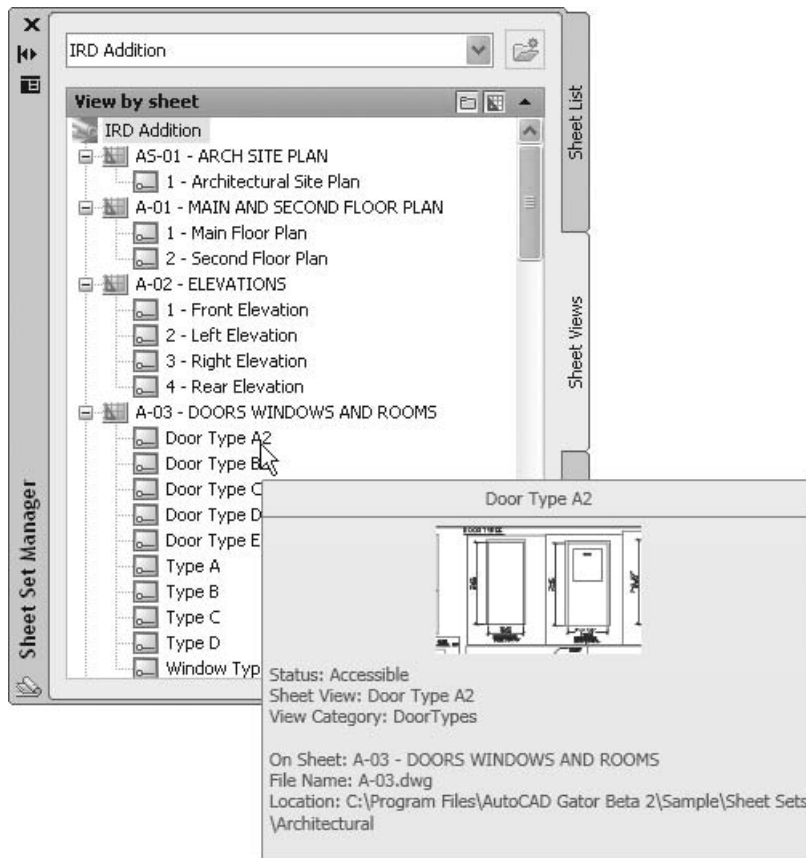


- While you are opening the **Sheet Set Manager** palette, you can do the following:
  - When you are in the **Sheet List** tab, if you point the mouse to one of the sheets you will see the following:



- This box displays a preview of the sheet and complete information such as: **Status**, name and number, **Description**, **File Name**, and so on.
- If you double-click any sheet, it will open and you will be able to change it as you wish.

- While you are in the **Sheet Views** tab, if you point the mouse to one of the views you will see the following:

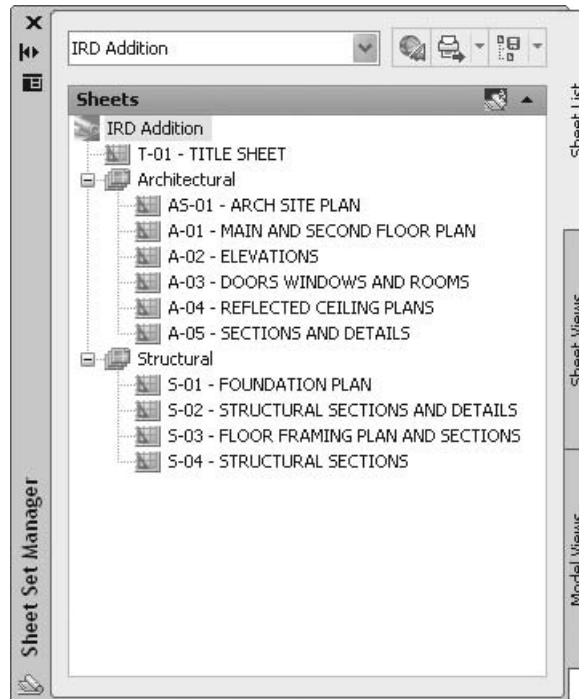


- This box displays the view and the complete information about it.
- If you double-click the name of the view, it will open the sheet containing this view. When you move your mouse outside the palette it will zoom to it.

## 15.4 SHEET SETS AND FILES

- A very important fact should be included here: The DST file contains only the associations and information that define the sheet set.
- When you create a new sheet set, AutoCAD automatically creates a new folder in the “My Documents” folder in your computer. It is highly recommended that you change the folder containing the sheet set file.

- Let's return to the example we discussed in the previous pages:



- As you can see from the preceding example, there are 11 sheets. One is linked directly to the sheet set, six sheets are located under the “Architectural” subset, and four under the “Structural” subset.
- AutoCAD created a file for each sheet; you can name the file whatever you want. In our example, the name of the file is the same as the number of the sheet.
- Each file contains one layout. The name of the layout is the same as the name of the sheet (e.g., ARCH SITE PLAN is the name of the layout in the file AS-01).

- If you go to the “My Documents” folder and browse until you reach the exact folder in which you stored your sheet set, you will see something like the following:



- If you count the DWG files, you will find 11, which is the exact number of sheets in the **Sheet Set Manager**.

## WORKING WITH THE SHEET SET MANAGER (GENERIC)



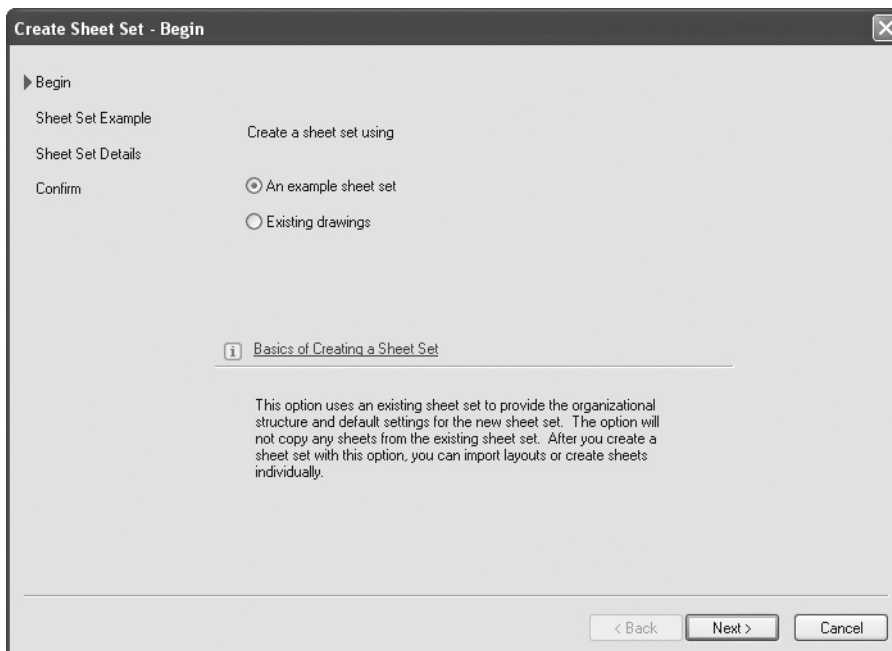
### Exercise 80

- Start AutoCAD 2010.
- Close all files.
- Start the **Sheet Set Manager**. From the pop-up list, select **Open**.
- Go to the folder named: \AutoCAD 2010\Sample\Sheet Sets\Manufacturing.
- Select the file *manufacturing sheet set.dst*.
- Double-click on the first sheet: **01 – Cover Sheet**.
- Zoom to the lower right-hand corner of the layout. You will see the name of the sheet and the sheet number as fields.
- Close the file without saving it.
- Point your mouse to the second sheet. You will see a preview and more information about the file.
- Go to the **Sheet Views** tab.
- Double-click the view **D-D** and see how the file will open. Next, zoom to the view.
- Close the file without saving it.

- Go to the **Model Views** tab and click the plus sign on the left-hand side of the only folder available.
- You will see the list of the files containing the views that will be imported as **XREFs** in the sheets.
- Using file *VW252-02-0300.dwg*, double-click the view **Detail A**. AutoCAD will open the file and zoom to the view.
- Close the file without saving it.
- Close the sheet set.

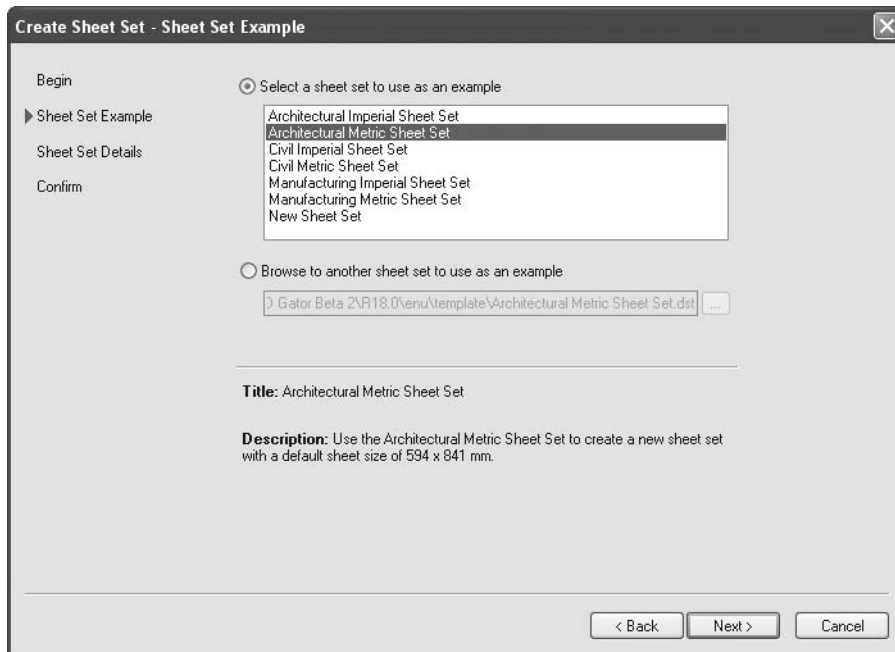
## 15.5 CREATING A NEW SHEET SET USING AN EXAMPLE

- As mentioned earlier, there are two ways to create a sheet set:
  - Using an example sheet set
  - Using existing drawings
-  Before we continue, please note the following:
  - We will assume from now on that the **Sheet Set Manager** palette is open.
  - You cannot create a new sheet set unless you have at least one file open.
- To create a new sheet set, use the pop-up list at the upper left-hand part of the **Sheet Set Manager**. Select the **New Sheet Set** option.
- The following dialog box will appear:



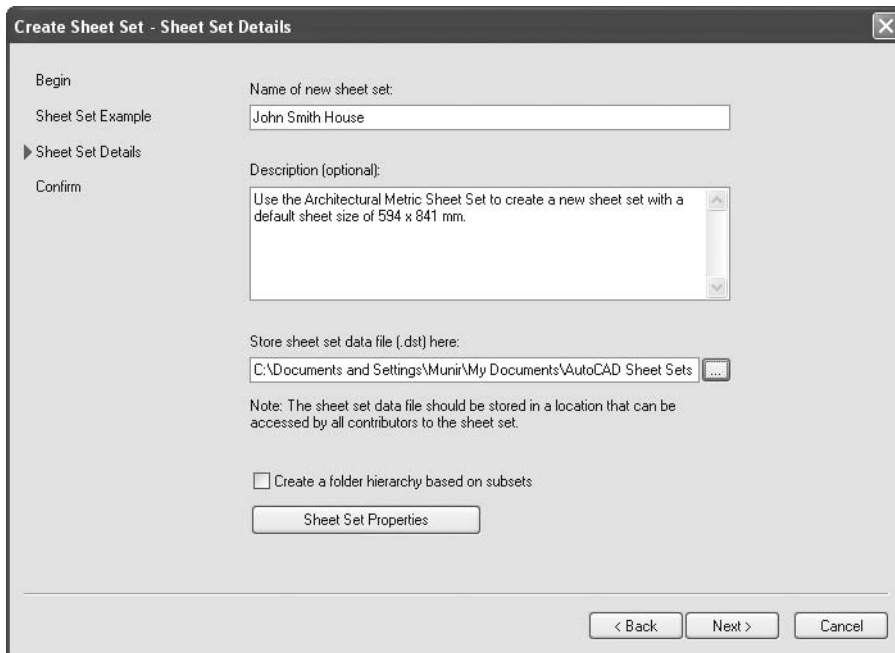


- Make sure that **An example sheet set** is already selected, and then click **Next**.
- The following dialog box will appear:



- Select one of the examples provided (a brief description will appear at the bottom of the dialog box).

- Remember that AutoCAD will only provide the subsets. No sheets will be added, as this is your responsibility. Click **Next** and the following dialog box will appear:



The dialog box is titled "Create Sheet Set - Sheet Set Details" and features a sidebar on the left with four steps: "Begin", "Sheet Set Example", "Sheet Set Details" (which is selected and expanded), and "Confirm". The main area contains the following fields and options:

- Name of new sheet set:** A text box containing "John Smith House".
- Description (optional):** A text area containing the text: "Use the Architectural Metric Sheet Set to create a new sheet set with a default sheet size of 594 x 841 mm."
- Store sheet set data file (.dst) here:** A text box showing the path "C:\Documents and Settings\Munir\My Documents\AutoCAD Sheet Sets" with a browse button (...).
- Note:** The sheet set data file should be stored in a location that can be accessed by all contributors to the sheet set.
- ☐ Create a folder hierarchy based on subsets
- Sheet Set Properties** button

At the bottom right, there are three buttons: "< Back", "Next >", and "Cancel".

- Type in the name of the sheet set.
- Type in any description (if needed).
- Select the place to store the DST file (the default location is the "My Documents" folder).
- Decide whether you wish to create a folder hierarchy based on subsets.

- Click the **Sheet Set Properties** button. The following dialog box will appear:

**Sheet Set Properties - John Smith House**

| Sheet Set                 |                                                   |
|---------------------------|---------------------------------------------------|
| Name                      | John Smith House                                  |
| Sheet set data file       | C:\Documents and Settings\Munir\My Docume...      |
| Description               | Use the Architectural Metric Sheet Set to crea... |
| Model view                |                                                   |
| Label block for views     | Drawing Block Title(C:\Documents and Setting...   |
| Callout blocks            | Callout(C:\Documents and Settings\Munir\Loc...    |
| Page setup overrides file | C:\Documents and Settings\Munir\Local Settin...   |

| Project Control   |  |
|-------------------|--|
| Project number    |  |
| Project name      |  |
| Project phase     |  |
| Project milestone |  |

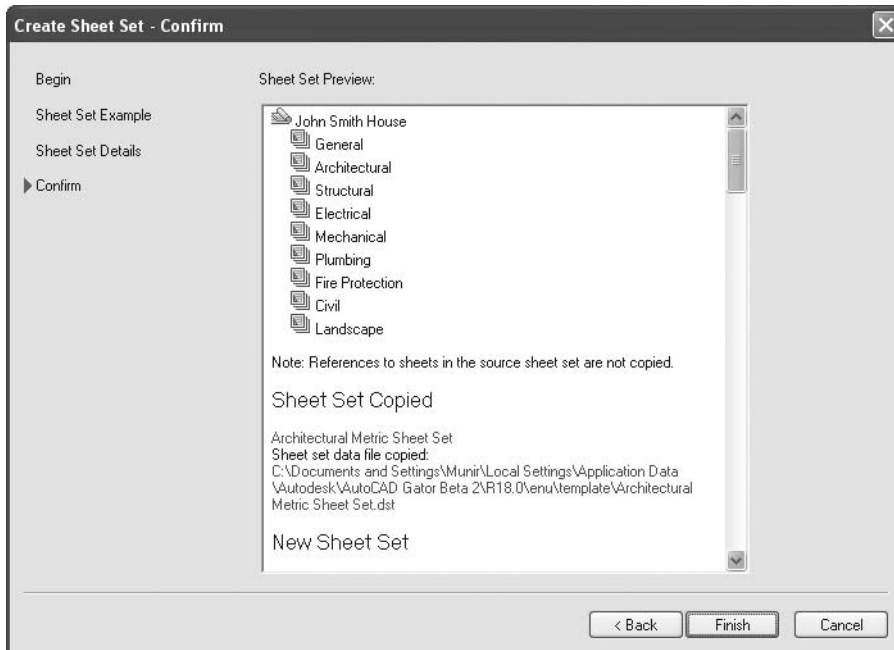
| Sheet Creation          |                                               |
|-------------------------|-----------------------------------------------|
| Sheet storage location  | C:\Documents and Settings\Munir\My Docume...  |
| Sheet creation template | ISO A1(C:\Documents and Settings\Munir\Loc... |
| Prompt for template     | No                                            |

| Sheet Set Custom Properties |       |
|-----------------------------|-------|
| Client                      | Value |
| Project Address Line 1      | Value |

Buttons: Edit Custom Properties..., OK, Cancel, Help

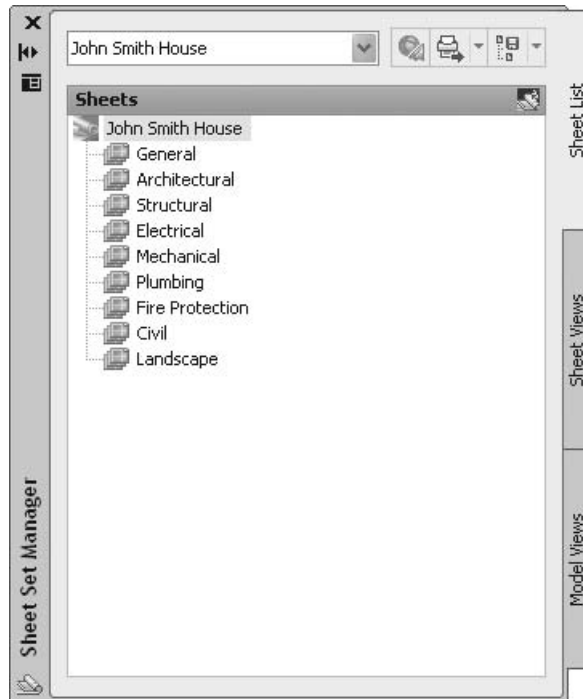
- You must specify several data here. We will cover the most important:
  - **Sheet storage location** is not the DST file, but the sheets themselves. It is highly recommended to keep it in the same place as the DST file.
  - **Sheet creation template** is where you should specify the path for the DWT file that all sheets will be based on.
  - **Project Control** and **Sheet Set Custom Properties** data is optional, but beneficial to the user if you fill it in.
- Click **OK**, and then click **Next**.

- The following dialog box will appear:



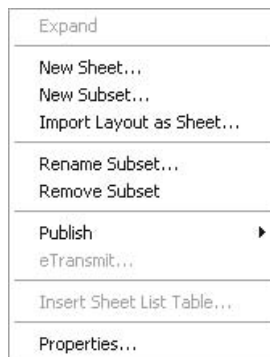
- Look at the preview, if you want to change anything, click **Back**. Otherwise, click **Finish**.

- You will see something similar to the following:

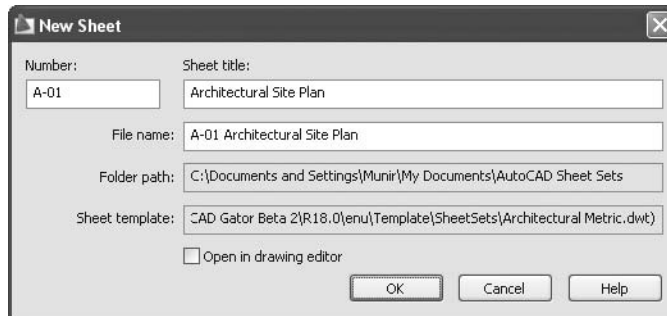


### Right-Click Subset

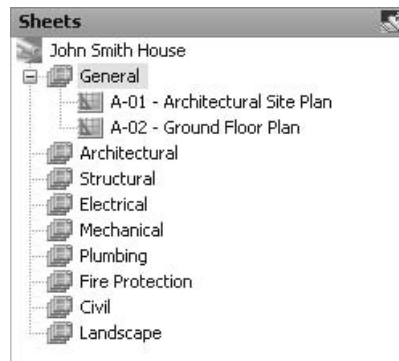
- As you can see, only subsets are visible.
- To create a new blank sheet inside a subset, perform the following steps:
  - Select the desired subset.
  - Right-click and the following menu will appear:



- Select the **New Sheet** option. The following dialog box will appear:



- Fill in the number, the sheet title, and the file name. Click **OK**.
- You will see something similar to the following:



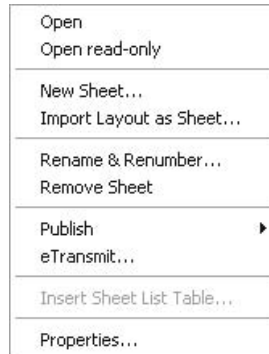
- If you double-click on any sheet, it will open and you will see that items came from the template file. The data that you enter in the **Sheet Set Properties** will be filled according to the sheet.



- Using the same menu, you can also create a **New Subset**, **Rename Subset**, and **Remove Subset**.

### Right-Click Sheet

- Right-click the desired sheet. The following menu will appear:



- You can do the following:
  - **Open** the sheet (double-clicking the name will also open it).
  - Create a **New Sheet** inside the same subset.
  - **Rename & Renumber** the sheet.
  - **Remove** the sheet.
- The other options will be discussed later.

## CREATING A SHEET SET USING AN EXAMPLE (GENERIC)



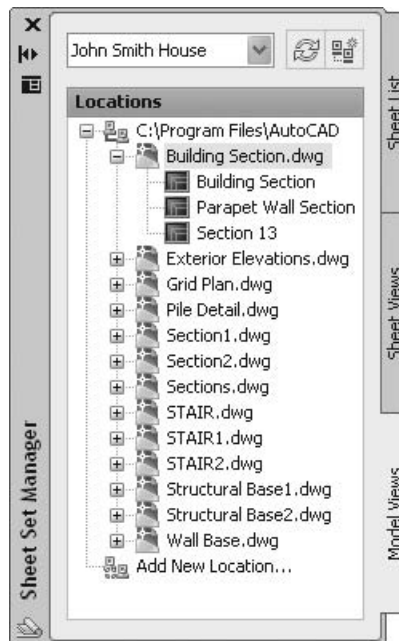
### Exercise 81

- Start AutoCAD 2010.
- Make sure that there is one file open.
- Start the **Sheet Set Manager** and create a new sheet set using an example sheet set.
- Use the **Architectural Imperial** example.
- The name of the new sheet set is **John Smith House**.
- Store the sheet set in the following folder: Exercises\Sheet Set\John Smith House.
- Click **Finish** to finish the creation of the sheet set.
- Go to the **Model Views** tab, click **Add New Location**, and find the “John Smith House Drawings” folder.
- Go back to the **Sheet List** tab and remove the **General** subset.
- Under **Architectural** create four new sheets and name them:
  - **A-01 Ground Floor – Overall**
  - **A-02 Ground Floor – Architectural Details**
  - **A-03 First Floor – Overall**
  - **A-04 First Floor – Architectural Details**

- Under **Structural** create a sheet **S-01** titled **Column Layout**.
- Save the file and close it.

## 15.6 ADDING VIEWS

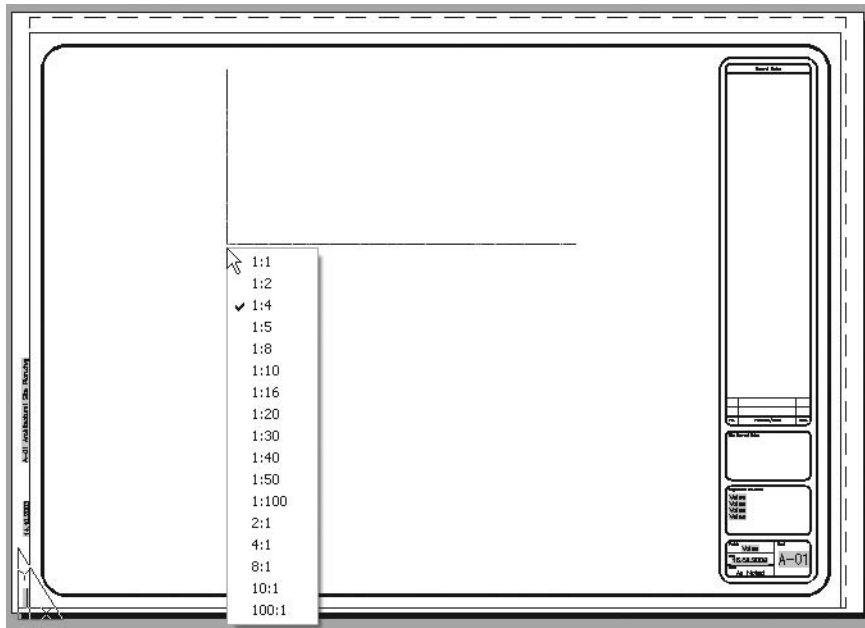
- Up until now, there are no links between the drawings in your project (which you have already finished) and the sheet set.
- To create this link you should add model views that have already been created in your drawings as blank sheets.
- These will be imported as an **XREF** to the original files.
- You must scale each view before inserting them.
- Perform the following steps:
  - Make sure you are in the **Model Views** tab.
  - Double-click **Add New Location**.
  - A dialog box will appear to select the folder your files reside in.
  - You will see something like the following:



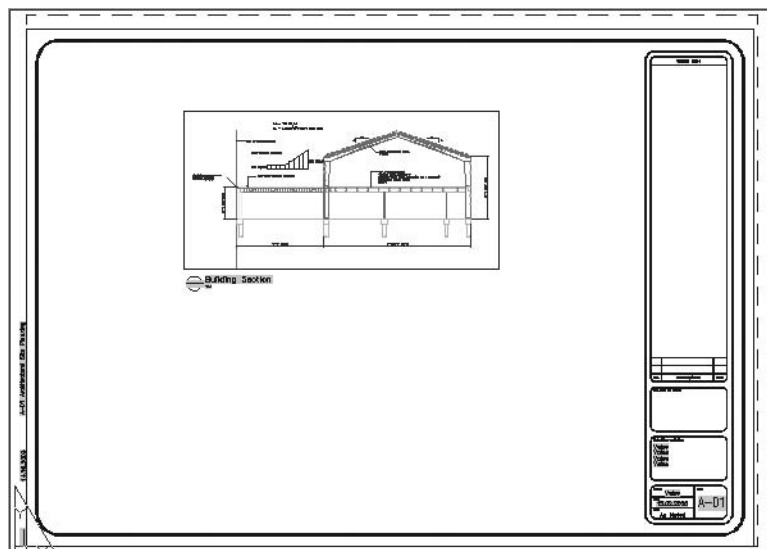
- If there is a plus sign left of the file, then it means there are named views in this file. Expand the file to see a listing of these views.
- Select the desired view, then drag and drop inside the desired sheet.



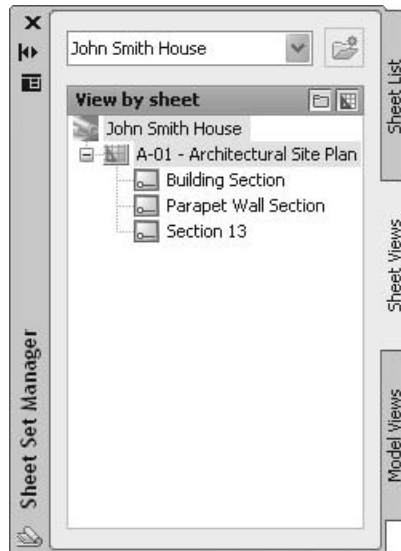
- You must specify the insertion point. Before you do, right-click, and you can choose from a list of the scale factors available. See the following illustration:



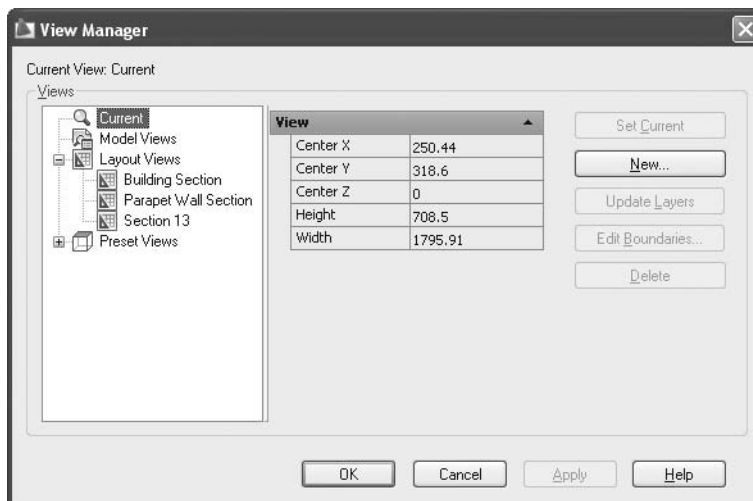
- Select the desired scale factor and specify the insertion point.
- You will get something like the following:



- Once you are done, take a look at the **Sheet Views** tab. You will see something like the following:



- Although there is more than one sheet in this sheet set, only one is listed (this contains the views).
- If you start the **View** command, you will see the following:



---

## ADDING VIEWS INTO SHEETS (IMPERIAL)

---



### Exercise 82

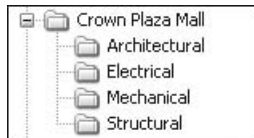
- Start AutoCAD 2010.
- Open the sheet set we created in Exercise 81.
- Double-click the **A-01** sheet; the file will be open.
- Go to the **Model Views** tab and click the plus sign beside *Ground Floor.dwg*.
- Click and drag the **Overall** view. First, right-click and select 1'=1'.
- Insert the other four views in **A-02** using 1'=1'.
- Insert the **Overall** view from *First Floor.dwg* into **A-03** using 1'=1'.
- Insert the other two views in **A-04** using 1'=1'.
- Insert the only view in *Structural.dwg* into **S-01** using 1'=1'.
- Save and close the **John Smith House** sheet set.

---

## 15.7 CREATING A NEW SHEET SET USING EXISTING DRAWINGS

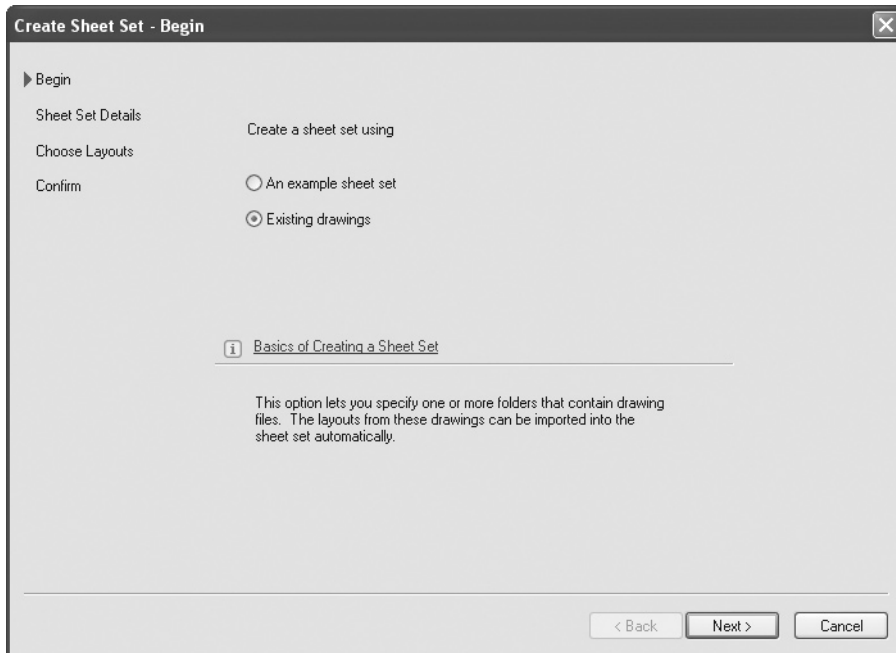
---

- If you want to create a new sheet set using an existing drawing, you should read the following guidelines:
  - Arrange your files by creating a hierarchy of folders. See the following example:

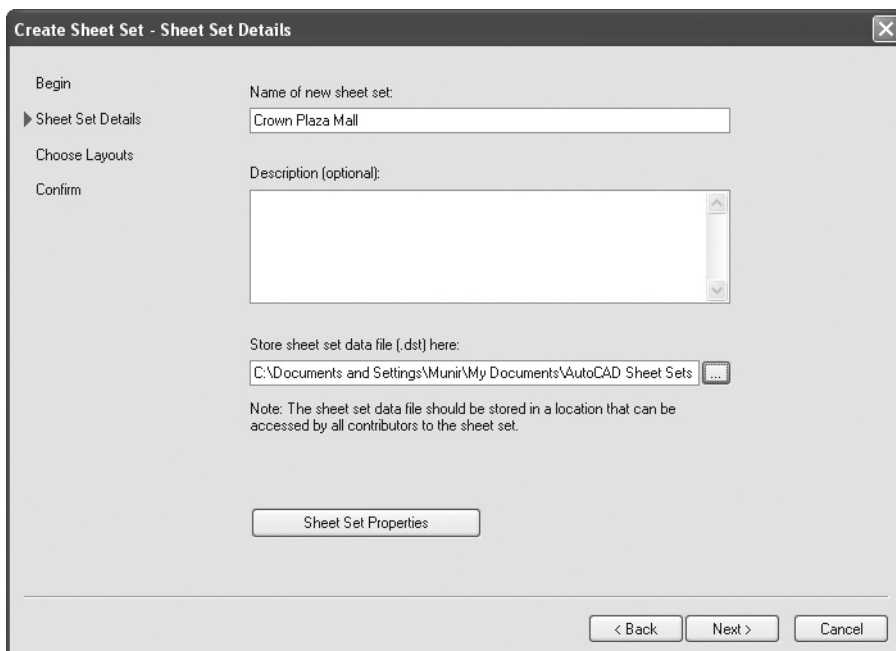


- Save all of your drawings within the respective folder.
- Make sure that each file only contains one layout. It is permissible to have multiple layouts to be converted to sheets in the same file, but if you plan to create your sheet set over a network and more than one person will be involved, Autodesk® recommends that you keep a single layout in a file to avoid issues.
- The sheet set will convert each folder to a subset and each layout in each file will be converted to a sheet.

- Select the **New Sheet Set** option. The following dialog box will appear:



- Select the **Existing drawings** option and click **Next**. The following dialog box will appear:



- Type in the name of the sheet set.
- Type in any description (if needed).
- Select the place to store the DST file (the default location is the “My Documents” folder).
- Click the **Sheet Set Properties** button. The following dialog box will appear:

| Sheet Set                 |                                                 |
|---------------------------|-------------------------------------------------|
| Name                      | Crown Plaza Mall                                |
| Sheet set data file       | C:\Documents and Settings\Munir\My Docume...    |
| Description               |                                                 |
| Model view                |                                                 |
| Label block for views     |                                                 |
| Callout blocks            |                                                 |
| Page setup overrides file | C:\Documents and Settings\Munir\Local Settin... |

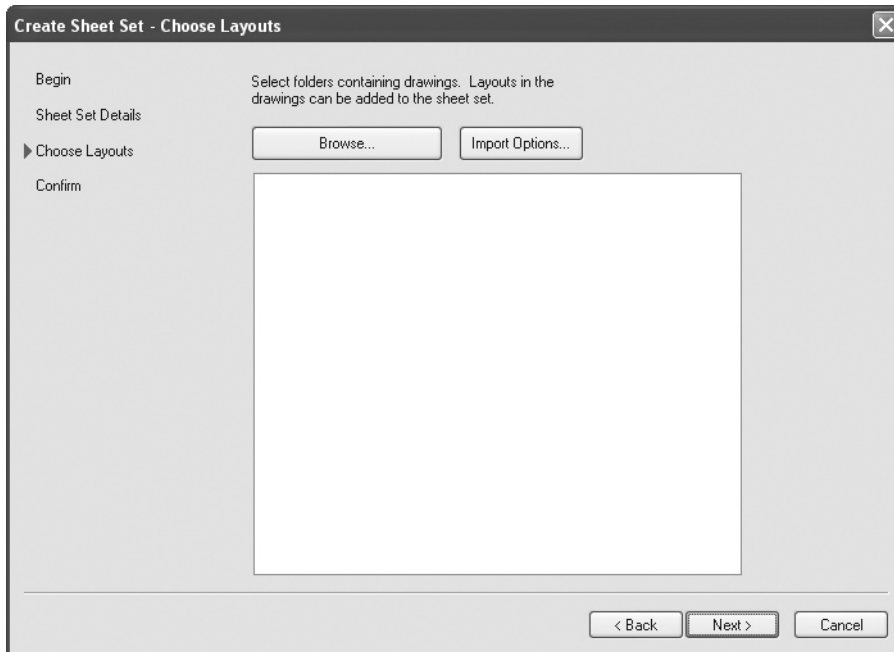
| Project Control   |  |
|-------------------|--|
| Project number    |  |
| Project name      |  |
| Project phase     |  |
| Project milestone |  |

| Sheet Creation          |                                               |
|-------------------------|-----------------------------------------------|
| Sheet storage location  | C:\Documents and Settings\Munir\My Docume...  |
| Sheet creation template | Arch D(C:\Documents and Settings\Munir\Loc... |
| Prompt for template     | No                                            |

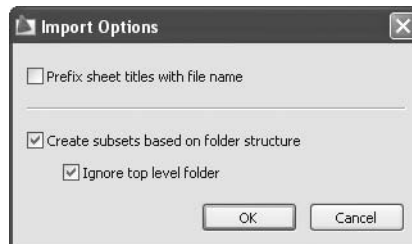
Buttons: Edit Custom Properties..., OK, Cancel, Help

- You must specify some data here. We will cover the most important data required: **Sheet storage location**, **Sheet creation template**, and **Project Control**.
  - **Sheet storage location:** It is highly recommended to keep the sheets in the same place as the DST file.
  - **Sheet creation template:** You should specify the path for the DWT file that the new sheets will be based on.
  - **Project Control:** This data is optional, but beneficial for the user if it is filled in.

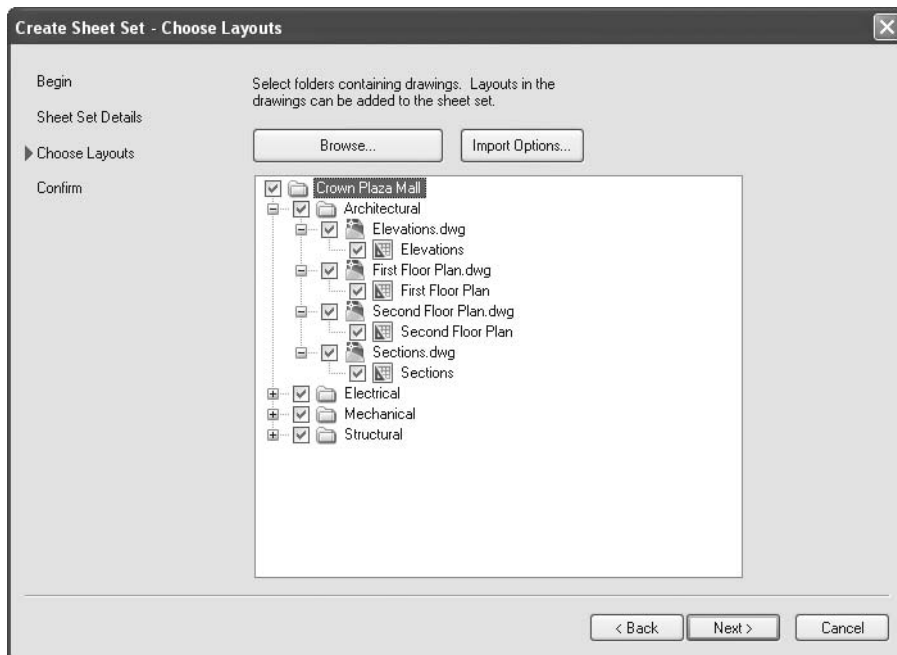
- Click **OK**, and then click **Next**. The following dialog box will appear:



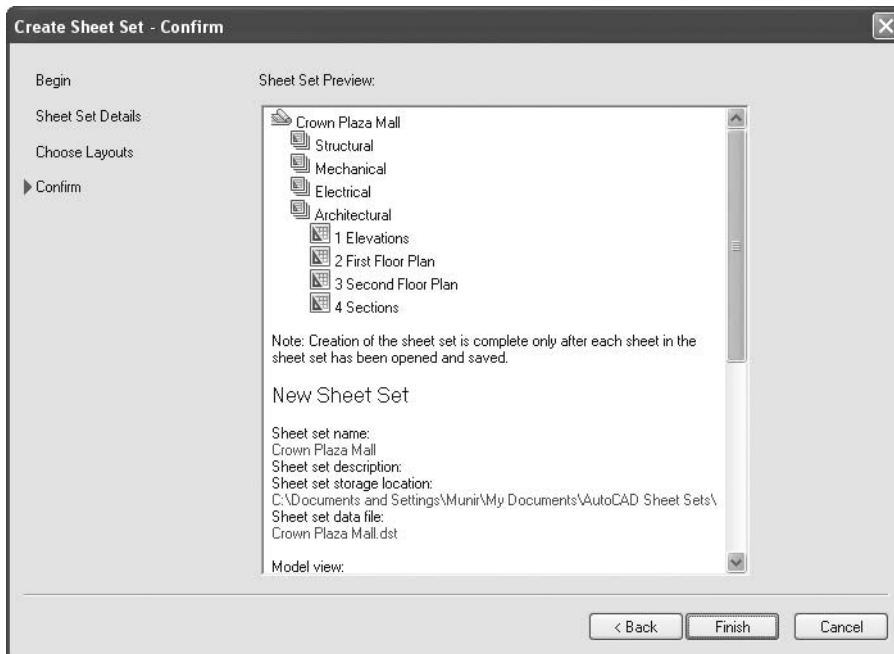
- Before you browse to the desired folder, check the **Import Options** button. You will see the following dialog box:



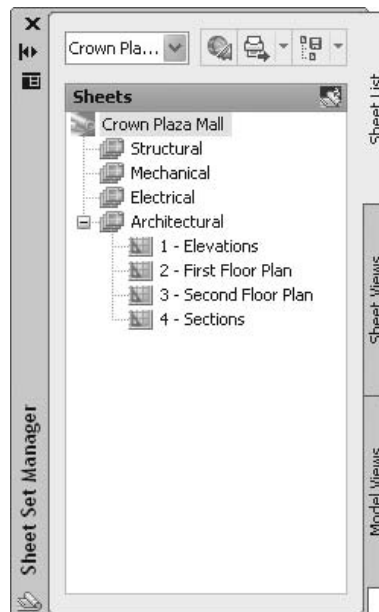
- For the **Import** options, you can:
  - **Prefix sheet titles with file name**
  - **Create subsets based on folder structure**
  - **Ignore top level folder**
- After you finish the **Import** options, click the **Browse** button and select the folder or folders that contain the desired files. You will see something like the following:



- By default, all files and all layouts are selected. You can deselect any of these. When you are finished, click **Next**. You will see the following dialog box. Click **Finish** or **Back**.

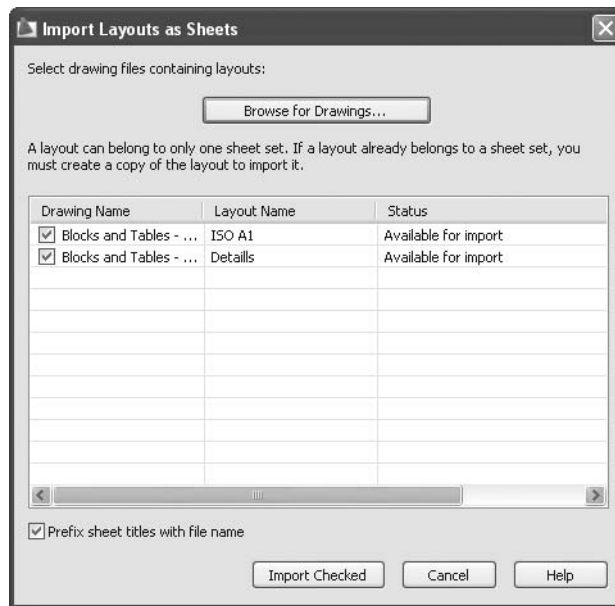


- The following will be shown in the **Sheet Set Manager**:



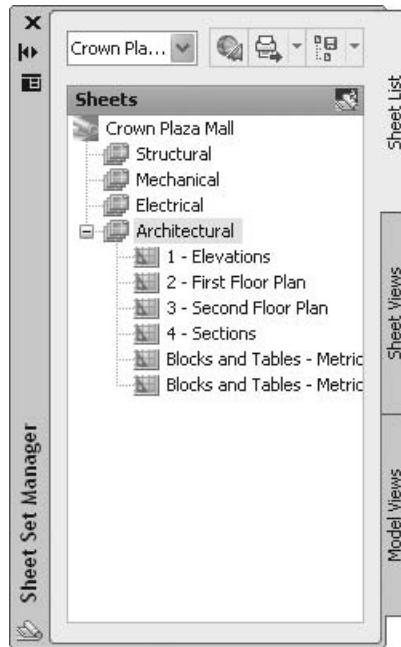


- You can do the following:
  - Rearrange the subsets and sheets by the drag-and-drop method.
  - Rename subsets, and renumber and rename sheets.
  - Create new subsets.
  - Create new sheets.
- Also, you can **Import Layout as Sheet** by performing the following steps:
  - Select the desired subset, right-click, and select **Import Layout as Sheet**. The following dialog box will appear:



- Click the **Browse for Drawings** button to search for the desired drawing(s).
- Select the drawings.
- Click the **Import Checked** button.

- The **Sheet Set Manager** will look like the following:



## CREATING SHEET SETS USING EXISTING DRAWINGS (GENERIC)



### Exercise 83

- Start AutoCAD 2010.
- Make sure that there is one file open.
- Start the **Sheet Set Manager** and create a new sheet set using existing drawings. Name it **Florida Villa** and save the DST file in: Exercise folder\Sheet Set\Florida Villa.
- Click the **Import Options** button, and check the following:
  - **Prefix sheet titles with file name** is turned off.
  - **Create subsets based on folder structure** is turned on.
  - **Ignore top level folder** is turned on.
- Click the **Browse** button and select the drawings in: Exercise folder\Sheet Set\Florida Villa.

- Click the plus sign near **Structural** and turn off the only sheet.
- Click **Finish** to end the command.
- Click the first sheet in the list and rename it to be **A-01 Ground Floor – Overall**.
- Perform the following steps:
  - Rename the *third* sheet to be **A-01 Ground Floor – Overall**.
  - Rename the *fourth* sheet to be **A-02 Ground Floor – Architectural Details**.
  - Rename the *first* sheet to be **A-03 First Floor – Overall**.
  - Rename the *second* sheet to be **A-04 First Floor – Architectural Details**.
- Rearrange them to be sorted by the number of sheets.
- Create a new subset and call it **Structural**.
- Right-click the new subset and select the **Import Layout as Sheet** option.
- Select **Browse for Drawings**, and select those in: Exercise folder\Sheet Set\Florida Villa Drawings\Structural. Next, choose *Structural.dwg*. You will find a single layout. Click **Import checked**.
- Rename the sheet to **S-01 Column layout**.
- Save and close the **Florida Villa** sheet set.

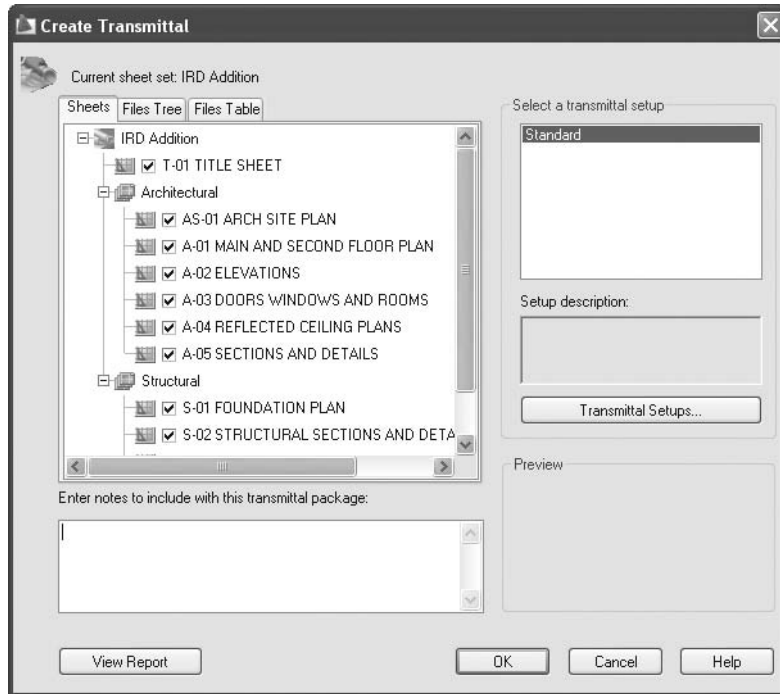
## **15.8 eTRANSMIT, ARCHIVE, AND PUBLISH SHEET SETS**

---

- We discussed and highlighted the advantages of the **eTransmit** command in Chapter 14.
- You can use the **eTransmit** command with sheet sets as well.
- According to the AutoCAD manual, eTransmit is for Internet transmittal, so why is there another command called **Archive**? According to the manual, you can use **Archive** for your internal work.
- We believe that one command is enough; **eTransmit** will do the job for both Internet and internal work.

**eTransmit**

- Using the **Sheet Set Manager**, select the name of the sheet set and right-click. Select the **eTransmit** option and the following dialog box will appear:

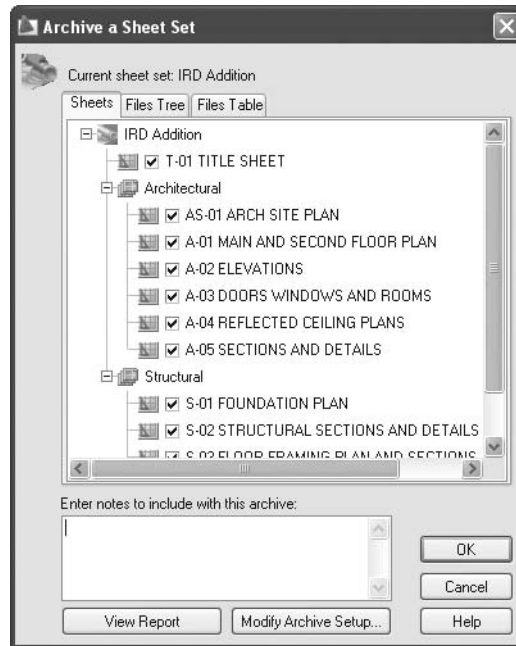


- You will find a new tab here called **Sheets**, which will show how the subsets and sheets are related to each other.
- The rest is the same as the **eTransmit** command already discussed.
- AutoCAD will save the DST file with the transmittal package.

 NOTE

### Archive

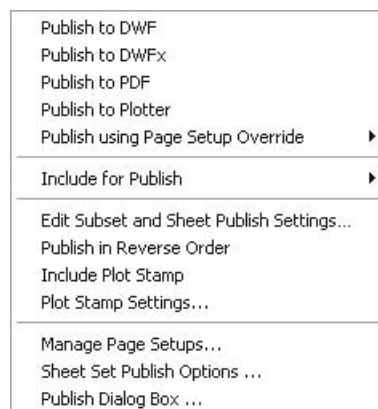
- Using the **Sheet Set Manager**, select the name of the sheet set and right-click. Select the **Archive** option and you will see the following dialog box:



- It is identical to the **eTransmit** dialog box, except that with this command, you cannot save the **Archive** setup, but you can change it if you wish.

### Publish

- Selecting the **Publish** option will present the following menu:



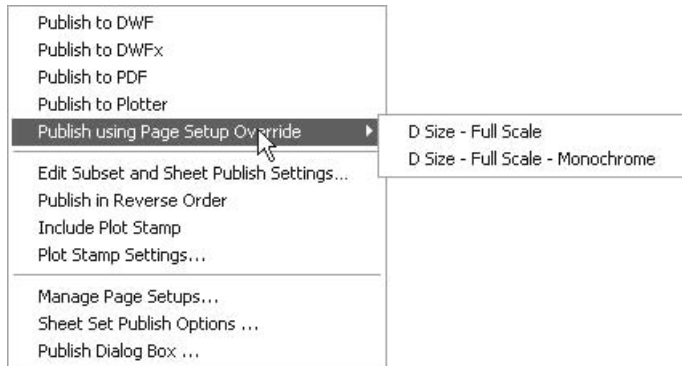
- These are the options included:
  - **Publish to DWF**, which allows you to create a multi-sheet DWF file for all the sheet sets.
  - **Publish to DWFx**, which is compatible with Microsoft® XPS reader. Therefore, you don't need **Autodesk Design Review** to open the file.
  - **Publish to PDF**, which allows you to create a PDF file for all the sheet sets.
  - **Publish to Plotter**, which allows you to send each sheet to the plotter configured inside the page setup.
  - **Publish using Page Setup Override**, where by default, each sheet contains the page setup that will be used once the **Publish** command is issued. Alternatively, the user can create other page setups to override the default page setups. These should be configured inside the **Properties** dialog box, just like the following:

| Sheet Set                 |                                                 |
|---------------------------|-------------------------------------------------|
| Name                      | Crown Plaza Mall                                |
| Sheet set data file       | C:\Documents and Settings\Munir\My Docume...    |
| Description               |                                                 |
| Model view                |                                                 |
| Label block for views     |                                                 |
| Callout blocks            |                                                 |
| Page setup overrides file | C:\Documents and Settings\Munir\Local Settin... |

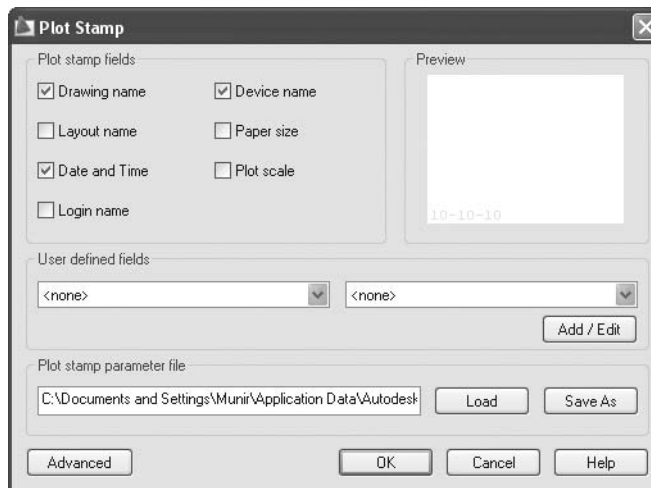
| Project Control   |  |
|-------------------|--|
| Project number    |  |
| Project name      |  |
| Project phase     |  |
| Project milestone |  |

| Sheet Creation          |                                               |
|-------------------------|-----------------------------------------------|
| Sheet storage location  | C:\Documents and Settings\Munir\My Docume...  |
| Sheet creation template | Arch D(C:\Documents and Settings\Munir\Loc... |
| Prompt for template     | No                                            |

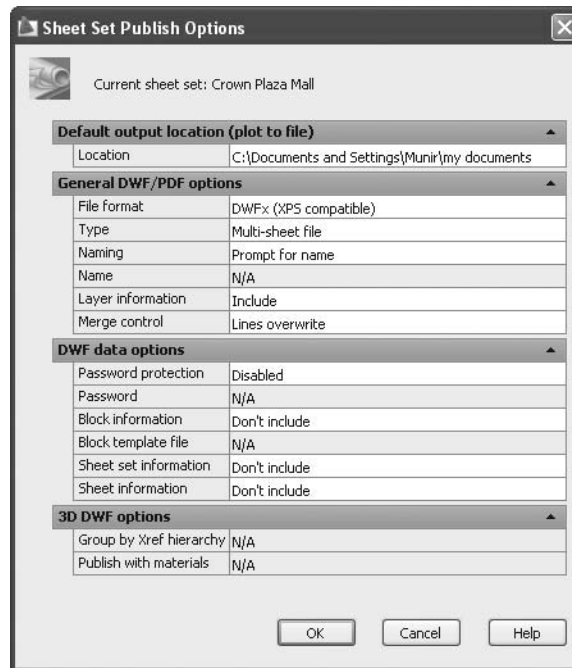
- You will see something like the following:



- **Publish in Reverse Order**, which is either turned on or off. By default, the publishing process will start with the first sheet. If you select this option, it will start with the last sheet.
- **Include Plot Stamp**, which is either turned on or off. By default, the **Plot Stamp** is not included.
- **Plot Stamp Settings**, which opens a dialog box that will help you set up your requirements for the desired stamp to show up on all printed sheets. You will see something like the following:



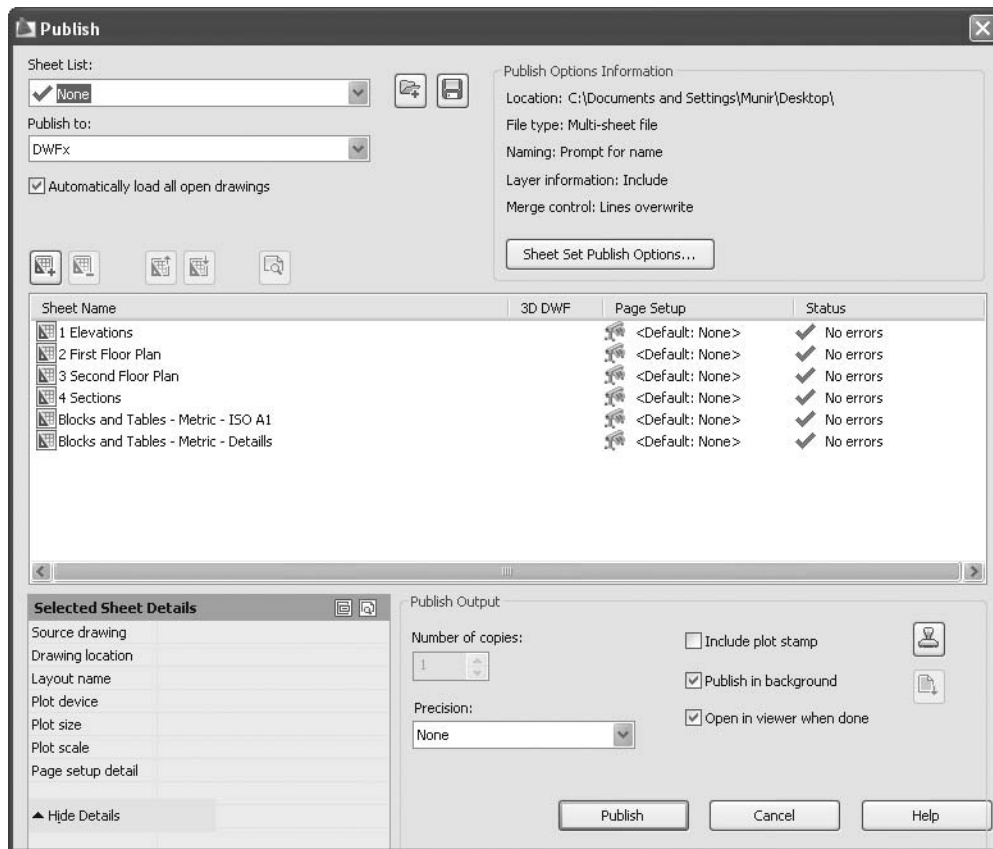
- **Manage Page Setups**, which will show you the **Page Setup Manager** that was thoroughly discussed earlier.
- **Sheet Set Publish Options**, which is identical to the **Publish** options of the **Publish** command. You will see the following dialog box:



- Be sure to include all of the necessary information inside the DWF file such as the **Layer**, **Sheet set**, **Sheet information**, and **Block information**.



- **Publish Dialog Box**, which will show you something like the following:



## eTRANSMIT, ARCHIVE, AND PUBLISH SHEET SETS (GENERIC)



### Exercise 84

- Start AutoCAD 2010.
- Make sure that there is one file open.
- Open the **Sheet Set Manager**.
- Open the **John Smith House** sheet set that you created earlier.

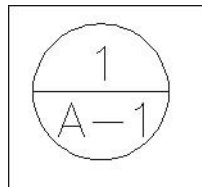
- Right-click the name of the sheet set and select the **eTransmit** option.
- Look at the sheets and files to make sure that all the necessary files have been included.
- Create an eTransmit package.
- Right-click the name of the sheet set and select **Publish/Publish Dialog Box**.
- Click the **Sheet Set Publish Options**, and make sure that **Sheet Set Information** and **Sheet Information** are both set to **Include**.
- Make sure that **Publish to** = DWF.
- Click **Publish** and save the DWF file in the same place as the DST file.
- Save and close the **John Smith House** sheet set.

## 15.9 LABEL BLOCKS AND CALLOUT BLOCKS

- When you are using an example and you add views to your blank sheets, a label block will be automatically added with some data relevant to the current sheet. You will see something like the following:



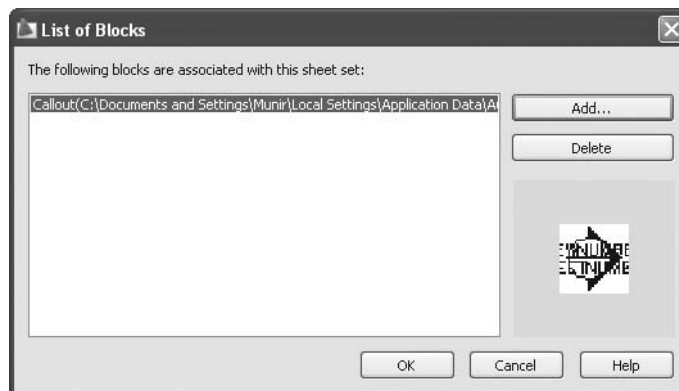
- The block contains the following data:
  - **View name**
  - **View number**
  - **Scale**
- All of these data are fields that will be input and updated by AutoCAD, not by the user.
- The callout block will look like the following:



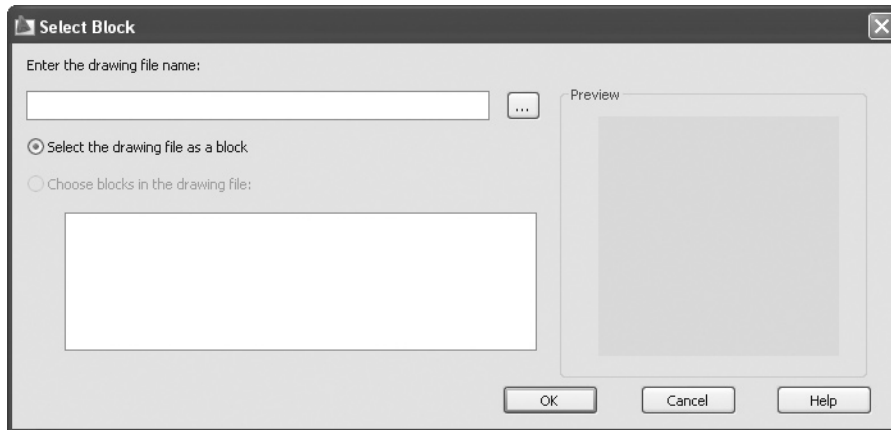
- The data included in this block are:
  - **View number** in this sheet
  - **Sheet number** containing the reference drawing
- All of the data are fields.
- You can specify the two types of blocks in the **Properties** dialog box of the sheet set. You have to specify whether the DWT or DWG contains both blocks:

| Sheet Set                 |                                                   |
|---------------------------|---------------------------------------------------|
| Name                      | John Smith House                                  |
| Sheet set data file       | C:\Documents and Settings\Munir\My Docume...      |
| Description               | Use the Architectural Metric Sheet Set to crea... |
| Model view                | C:\Program Files\AutoCAD\Gator Beta 2\Samp...     |
| Label block for views     | Drawing Block Title(C:\Documents and Setting...   |
| Callout blocks            | Callout(C:\Documents and Settings\Munir\Loc...    |
| Page setup overrides file | C:\Documents and Settings\Munir\Local Settin...   |

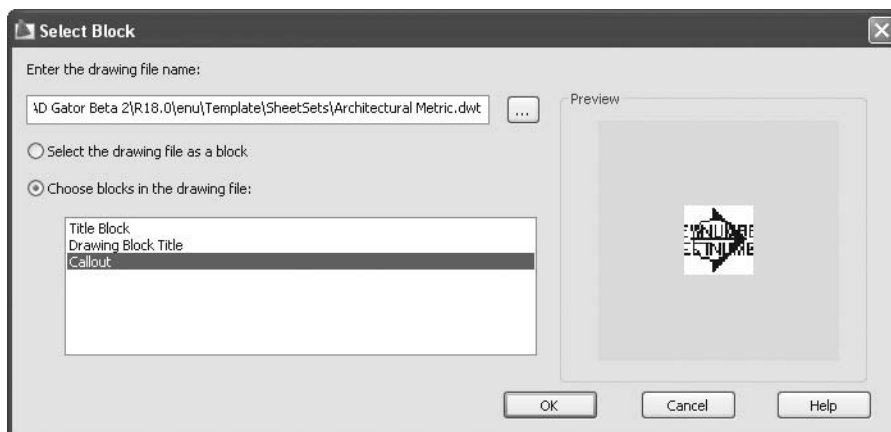
- Depending on the example sheet set you chose, AutoCAD will assign the file you get the two blocks from by default.
- AutoCAD comes with a folder called “Templates.” This folder contains some standard templates along with two more folders: one called “SheetSets,” which contains six templates for **Architectural**, **Civil**, and **Manufacturing**. The other folder contains two types of templates: one for **Metric** and the other for **Imperial**.
- If you picked **Architectural Metric** as an example, the following will be assumed:
  - **Sheet creation template** = Architectural Metric
  - **Label block for views** = Architectural Metric
  - **Callout blocks** = Architectural Metric
- If you want to use your own template for the blocks, click the small button with the three dots and you will see the following dialog box:



- Click the **Add** button. You will see the following dialog box:



- Click the small button with the dots inside it to select the desired file. You will see something like:



- You may specify other blocks in the current file or select the desired blocks from another file.
- You can create your own blocks if you wish and add all the desired fields you like.

---

**CHAPTER REVIEW**

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1. When you are creating a new sheet set using an example sheet set, you have to:
  - a. Create sheets
  - b. Create views in the files
  - c. Drag views as **XREFs**
  - d. All of the above
2. While you are bringing in a view from a file, you have to set the \_\_\_\_\_ of the view.
3. There are two methods used to create sheet sets in AutoCAD.
  - a. True
  - b. False
4. After you create a sheet set you can:
  - a. Archive it
  - b. Publish it using different file formats
  - c. Export it as DXF file
  - d. A and B
5. The label block for views and the callout block are preset and you cannot create your own.
  - a. True
  - b. False
6. If you would like to import another sheet from a layout in another file after you create your sheet set, you will right-click the desired subset, and select the \_\_\_\_\_ option.

---

**CHAPTER REVIEW ANSWERS**

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1. d
2. scale
3. a
4. d
5. b
6. **Import layout as Sheet**



# CAD STANDARDS AND ADVANCED LAYERS

## Chapter 16

### In This Chapter

- ◇ Introduction
- ◇ Creating a Standard File
- ◇ Configuring and Checking CAD Standards
- ◇ The **Layer Translator** Command
- ◇ Advanced Layers: General Look
- ◇ The **Property Filter**
- ◇ The **Group Filter**
- ◇ Right-Clicking the Filter
- ◇ The **Layer States Manager** Command
- ◇ The Settings Dialog Box
- ◇ Advanced Layer Functions

### 16.1 INTRODUCTION

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- CAD (and AutoCAD® in particular) is getting more complex as a new version is introduced each year with new commands and techniques.
- Companies using AutoCAD are employing many more CAD operators.
- Design by itself is getting more and more complicated.
- International bodies continue to issue new standards that address the new changes in design and how they will be delivered.
- Now companies—no matter how small—cannot continue to exist without CAD Managers. Without CAD Managers, companies will be without CAD standards, proper training, budgeting, and so on.

- One of the CAD Manager's duties is to establish CAD standards for his or her company by doing the following:
  - Adopting the right CAD standard for the industry his or her company is in.
  - Translating the standard inside AutoCAD into something tangible, such as **Layers**, **Dimension style**, **Text style**, **Table style**, **Layouts**, and so on.
  - Making sure that all CAD operators within the company are using these standards consistently.
  - Training all staff to use these standards efficiently.
  - Outlining the importance of these standards for the workflow and for the success of the project and the company.
  - Checking all drawings that are finished or partially developed to be sure that no standard violation took place.
- The AutoCAD **CAD Standard tool** enables CAD Managers to accomplish their duties swiftly.
- The large list of **Layer Advanced** functions allows the user to control any drawing regardless of the number of layers inside it.

---

## 16.2 CREATING A STANDARD FILE

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- Normally, each company has its own template files. These template files include all of the following:
  - **Layers** (naming convention, color, linetype, lineweight)
  - **Dimension styles**
  - **Multileader styles**
  - **Text styles**
  - **Table styles**
  - **Layouts** (including **Page Setup**, **Border block**, and **Viewports**)
- The best way to create a standard file is to open a template file and transform it to standard file (from a DWT file to a DWS file) using the **Save As** command.
- Standard files can only check the following:
  - **Layers** (any thing related to layers)
  - **Dimension styles**
  - **Text styles**
  - **Linetypes**

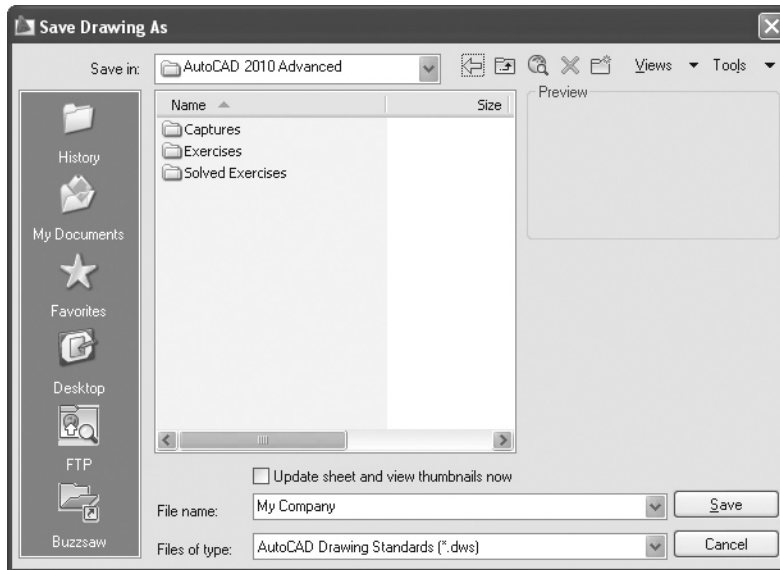


**Procedure**

- Create a new file using the desired template file. Using the **Applications Menu**, click **Save As** and select **AutoCAD Drawing Standards** as seen in the following:



- The following dialog box will appear:



- Type the name of the new standard file (\*.dws) and click the **Save** button.

## CREATING STANDARD FILES (GENERIC)



### Exercise 85

- Start AutoCAD 2010.
- Create a new file based on *Tutorial-mArch.dwt*, which comes with AutoCAD 2010.
- Go to the **Application Menu** and select **Save As/AutoCAD Drawing Standards**.
- Save the file in your “Exercise” folder and call it *My\_Company.dws*.
- Save the file and close it.

## 16.3 CONFIGURING AND CHECKING CAD STANDARDS

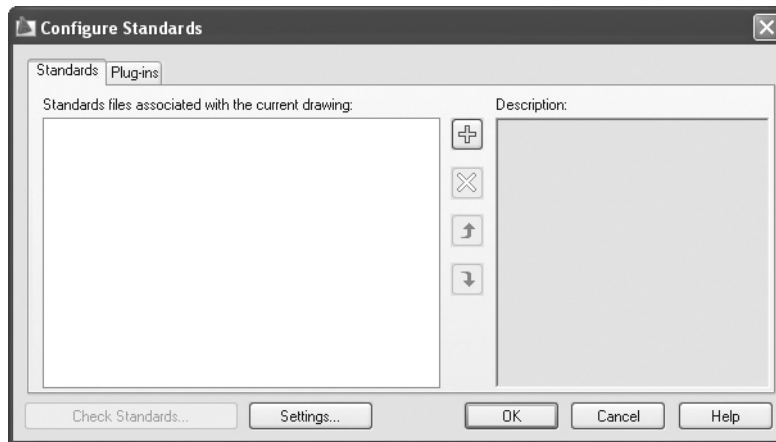
- You will create a standard file once.
- There are two more steps to perform in order to complete this process:
  - **Configuring** the standard file (linking your current DWG to an existing DWS file)
  - **Checking** the compliance of your DWG compared to the DWS file

### Configuring

- Open the desired DWG file to check its compliance.
- Make sure you are on the **Manage** tab on the **Ribbon** and, using the **CAD Standards** panel, click the **Configure** button.

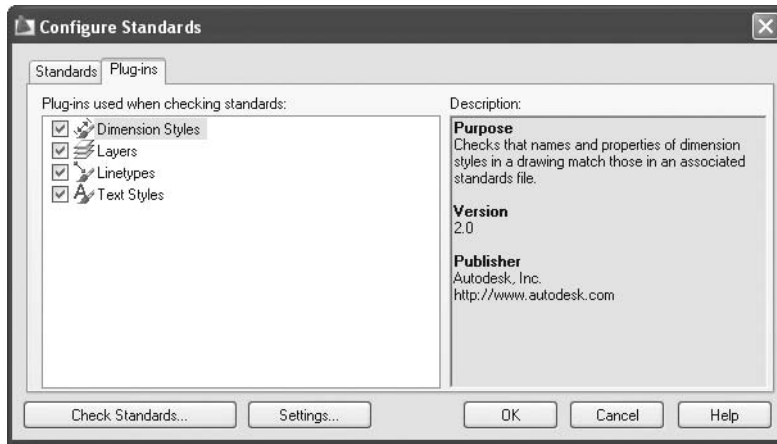


- The following dialog box will appear:



- Click the button with the plus sign in it. A dialog box will appear allowing you to select your DWS file. Select your desired file and click **OK**.

- Click the **Plug-ins** tab. You will see the following:



- Clear the unnecessary plug-ins that you do not want AutoCAD to check for.
- Click the **Settings** button. You will see the following dialog box:



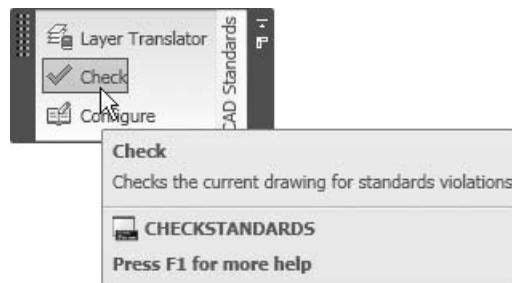
- Choose which **Notification** setting is suitable for you:
  - Disable standards notifications**
  - Display alert upon standards violation** (enabled while you are still working with the DWG file)
  - Display standards status bar icon** (see the following illustration)



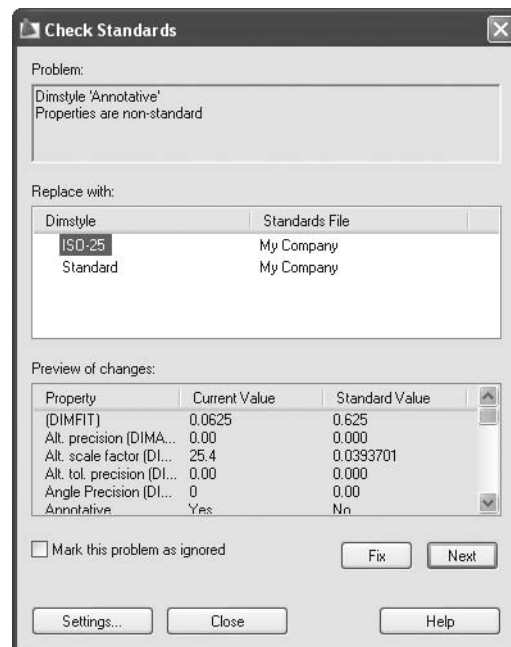
- Under **Check Standards settings**, set the following:
  - You can choose to **Automatically fix non-standard properties** or you can fix those properties manually.
  - You can either hide or **Show ignored problems**.
  - Select your **Preferred standards to use for replacements**.

### Checking

- Open the file you want to check for compliance.
- Make sure you are on the **Manage** tab on the **Ribbon** and, using the **CAD Standards** panel, click the **Check** button:



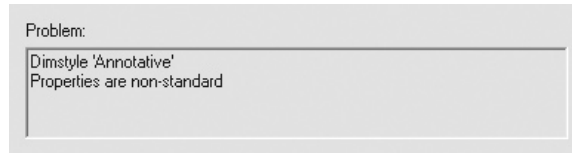
- The following dialog box will appear:



- There are three sections to be addressed:
  - **Problem**
  - **Replace with**
  - **Preview of changes**

#### Problem

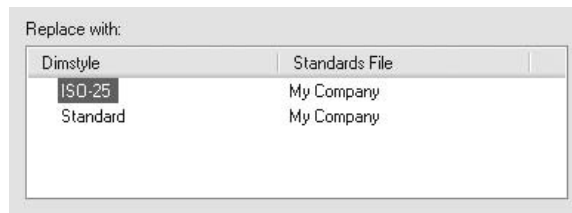
- If a problem is found, it will be listed in this section:



- In the preceding example, **Dimension style 'Annotative'** is not available in the standard file.

#### Replace with

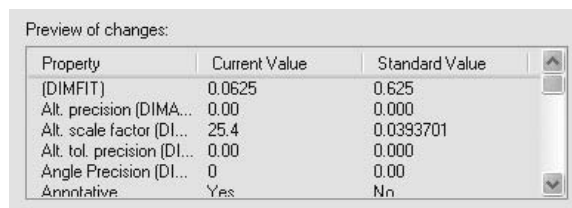
- AutoCAD will search the standard file and suggest a replacement for the existing style:



- In the preceding example, AutoCAD suggests either **ISO-25 Dimstyle** or **Standard Dimstyle** selected from the **Standards File**.
- Let's assume we choose **ISO-25**.

#### Preview of Changes

- AutoCAD will automatically show you that if you choose this replacement style the following changes will take place:



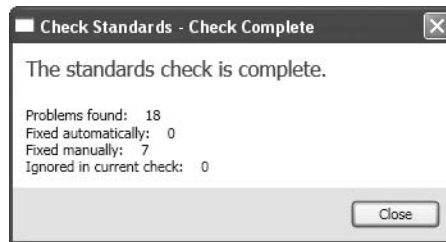
- In the preceding example, several modifications will take place. For one of these, **DIMFIT**, the **Current Value** is 0.0625 and the **Standard Value** is 0.625.

#### Mark This Problem as Ignored

- At the bottom left of the dialog box, you will see the following checkbox:

☐ Mark this problem as ignored

- Click this checkbox if you wish to accept this problem as is.
- You have two buttons to choose from: **Fix** and **Next**.
  - **Fix** allows you to fix the problem as the **Standards check** proposes.
  - **Next** allows you to ignore it this time, and the **Standards check** will show it as a problem again the next time.
- When you are done, you will see the following message:



- You can configure (link) the standard file to your file at any time.
- While you are working, if you created any change that violates the file standards, AutoCAD will alert you with the following message:



- If the **Standards check** finds a layer that does not exist in the standard file, but you choose to include it inside an existing layer in the standard file, what will happen to the old layer? AutoCAD will move all objects from the old layer to the new layer and delete the old layer from this file.

## CONFIGURING AND CHECKING STANDARDS (GENERIC)



### Exercise 86

- Start AutoCAD 2010.
- Open the *Exercise\_86.dwg* file.
- Start the **Configure** command and link the *My\_Company.dws* file to it.
- Go to **Plug-ins** and clear all the options except the **Layers** option.
- Check the right part of the **Status Bar** and make sure you can see the icon referring to the **Standard file**.
- Start the **Check** command and change the following:

| Layer Name      | Action                   |
|-----------------|--------------------------|
| A-Wall          | Replace with A-Walls     |
| A-Door          | Replace with A-Doors     |
| Partition       | Replace with A-Walls     |
| A-Window        | Replace with A-Windows   |
| Frame           | Replace with Title Block |
| A-Area          | Skip                     |
| Text            | Replace with Text        |
| A-Centerlines   | Skip                     |
| Centerline-TAGS | Skip                     |
| A-Layout        | Skip                     |
| A-Dimension     | Replace with Dimensions  |
| Staircase       | Skip                     |
| A-Furniture     | Skip                     |
| Vports          | Replace with Viewport    |

- Go to the **Layer Manager** tool palette to see how some of the layers have disappeared and others have been renamed.
- Save the file and close it.

## 16.4 THE LAYER TRANSLATOR COMMAND

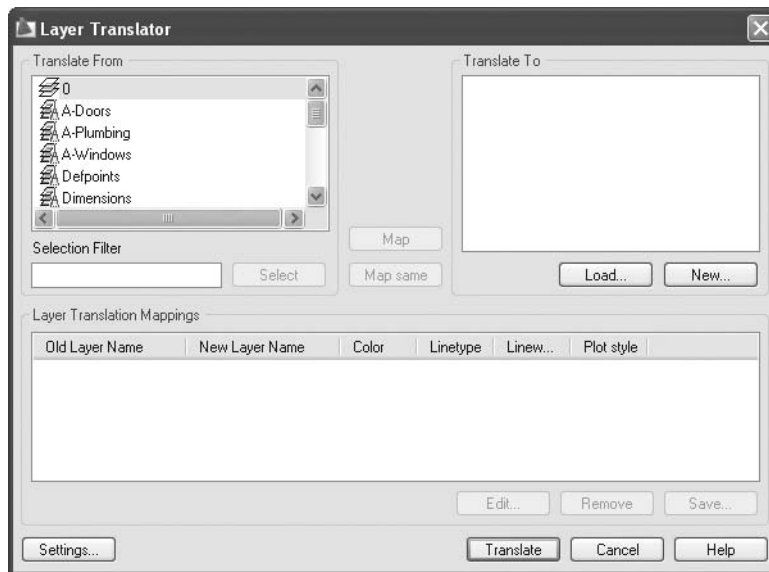
- When you are a CAD Manager you have full control and authority over the company's staff, but you don't have the same power over those outside the company.



- Assume you are with an AEC firm and you are receiving some DWG files as shop drawings from a contracting company. Your staff will want to check them and make some amendments. They are not happy with the contractor's naming conventions and choices of color.
- As a CAD Manager, you can translate the layers coming in these files into standards that your staff feels comfortable dealing with.
- Make sure you are on the **Manage** tab on the **Ribbon** and, using the **CAD Standards** panel, click the **Layer Translator** button:

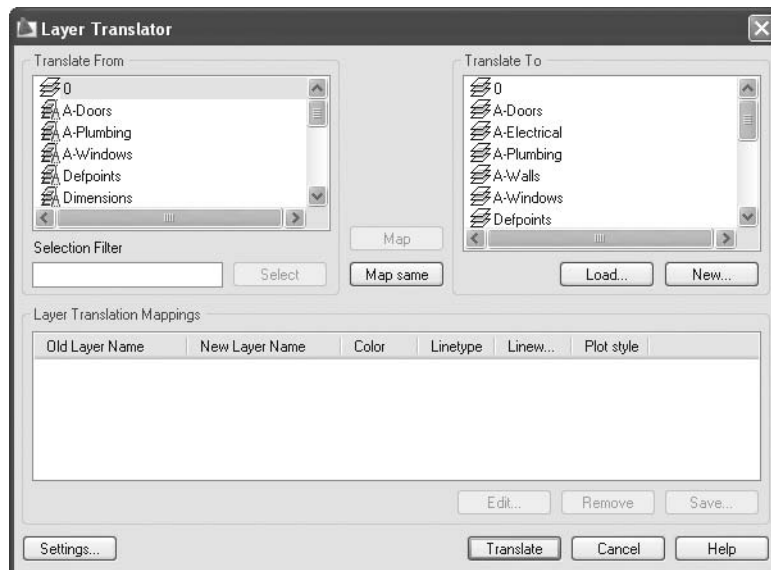


- The following dialog box will appear:



- Click the **Load** button to load the file containing your list of layers. You can have DWG, DWT, and DWS files loaded.

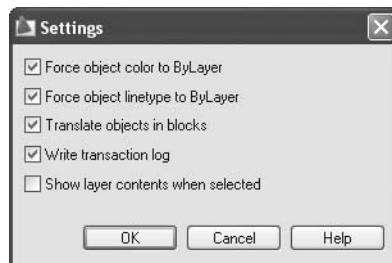
- After you load the file, you will see the following dialog box:



- If you click the **Map same** button, AutoCAD will find any layers with similar names. In order to exclude the layer from being mapped, you will see the following:



- Now, select a layer from the **Translate From** list and a layer from the **Translate To** list. Next, click the **Map** button. The layer will move from the left list and will appear in the list at the bottom of the window.
- Repeat until you are finished.
- You can save the list you have mapped for future mapping with similar files (i.e., files from the same company).
- You can click the **Settings** button at any time to show the following dialog box:



- When you are mapping, you can set any or all of the following:
  - **Force object color to ByLayer**
  - **Force object linetype to ByLayer**
  - **Translate objects in blocks**
  - **Write transaction log** (creates a log file of the translation process)
  - **Show layer contents when selected**
- Click **OK** when you are finished.

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## LAYER TRANSLATOR (GENERIC)



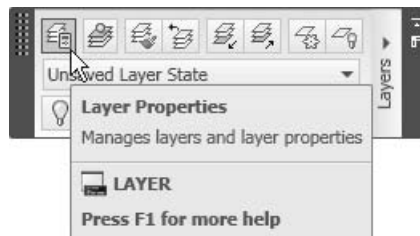
### Exercise 87

- Start AutoCAD 2010.
- Open the file *John Smith House Ground Floor Plan.dwg*.
- Start the **Layer Translator** command.
- Load *My\_Company.dws* as the file to translate to.
- Map the layers with the same name.
- Translate **Partition** layer to **A-Walls** layer.
- Do not translate the remaining layers.
- Save the translation as *My\_Layer\_Translation.dws*.
- Click the **Translate** button.
- Save the file and close it.

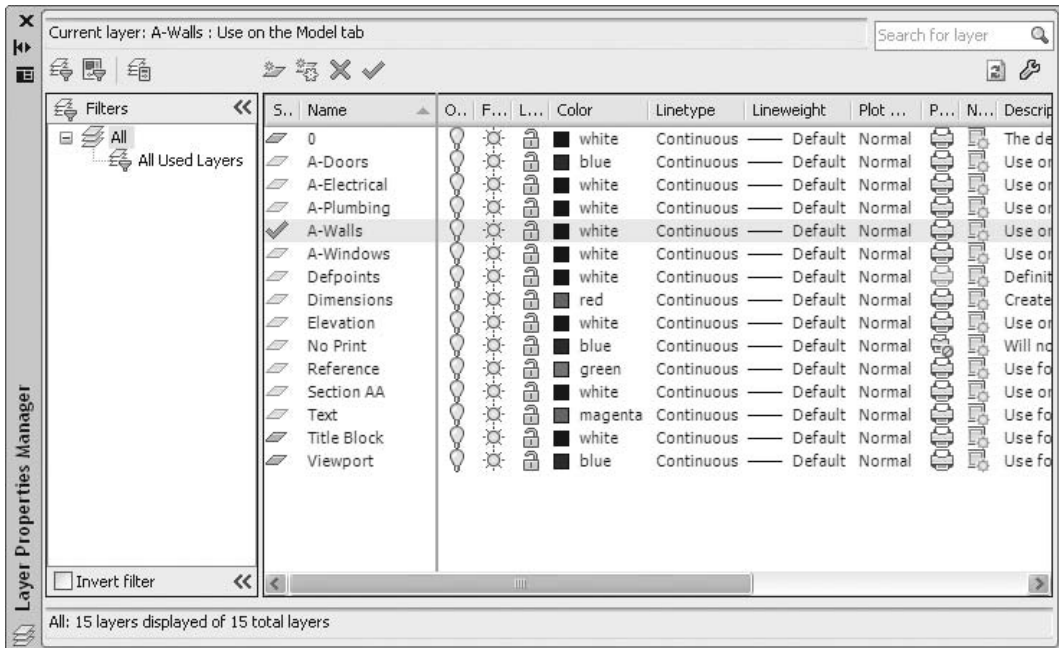
- Open the file *John Smith House First Floor Plan.dwg*.
- Start the **Layer Translator** command.
- Load the file *My\_Layer\_Translation.dws*.
- See how the layers were translated automatically?
- Click the **Translate** command.
- Save the file and close it.

## 16.5 ADVANCED LAYERS: GENERAL LOOK

- If you have a drawing with a large number of layers, use the **Property Filter** accessed through the **Layer Properties Manager** to create a filter based on a common property such as, **Name**, **Color**, **Freeze/Thaw**, **On/Off**, and so on.
- Also, you can create a group of layers with no common thread by using the **Group Filter**.
- You can create a **Layer State**, which can save the state of the current layers and restore them for future use.
- Make sure you are on the **Home** tab in the **Ribbon** and, using the **Layers** panel, click the **Layer Properties** button:



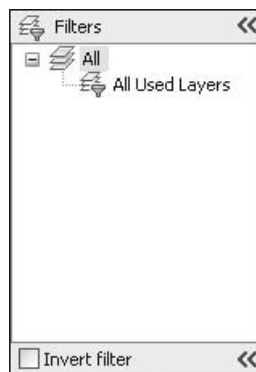
- The following palette will appear:



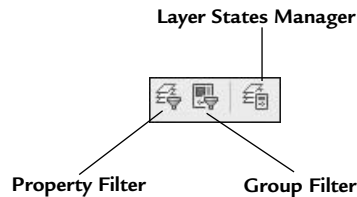
- Note the following:
  - At the lower left corner of the palette, you will see the total number of layers.

All: 15 layers displayed of 15 total layers

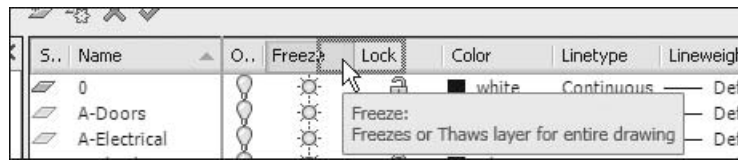
- At the left of the dialog box, you will see the **Filter** pane. By default, you will see something like the following:



- You can see there are two filters: **All** (includes all the layers used and all the unused layers) and **All Used Layers** (includes only used layers or non empty layers).
- The **Invert filter** checkbox allows you to select the opposite of the current filter.
- At the upper left-hand corner, you will see three buttons:



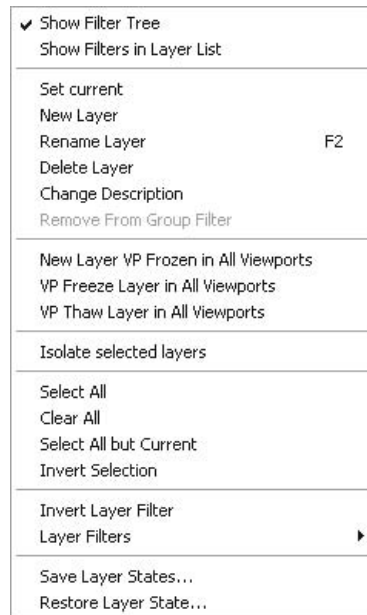
- In the preceding dialog box, you can also:
  - Change the place of a column using the drag-and-drop method.



- Sort the layers according to the column you choose (**Ascending** or **Descending**). In the following example, layers were sorted according to colors in ascending order.

| S.. | Name         | O.. | Freeze | Lock | Color   | Linetype   | Lineweight | Plot ... |
|-----|--------------|-----|--------|------|---------|------------|------------|----------|
|     | Dimensions   |     |        |      | red     | Continuous | Default    | Normal   |
|     | Reference    |     |        |      | green   | Continuous | Default    | Normal   |
|     | A-Doors      |     |        |      | blue    | Continuous | Default    | Normal   |
|     | No Print     |     |        |      | blue    | Continuous | Default    | Normal   |
|     | Viewport     |     |        |      | blue    | Continuous | Default    | Normal   |
|     | Text         |     |        |      | magenta | Continuous | Default    | Normal   |
|     | 0            |     |        |      | white   | Continuous | Default    | Normal   |
|     | A-Electrical |     |        |      | white   | Continuous | Default    | Normal   |
|     | A-Plumbing   |     |        |      | white   | Continuous | Default    | Normal   |
|     | A-Walls      |     |        |      | white   | Continuous | Default    | Normal   |
|     | A-Windows    |     |        |      | white   | Continuous | Default    | Normal   |
|     | Defpoints    |     |        |      | white   | Continuous | Default    | Normal   |

- Select any layer name and right-click. If you do so, you will see the following menu. There are many options available in this menu, all of which are self-explanatory.



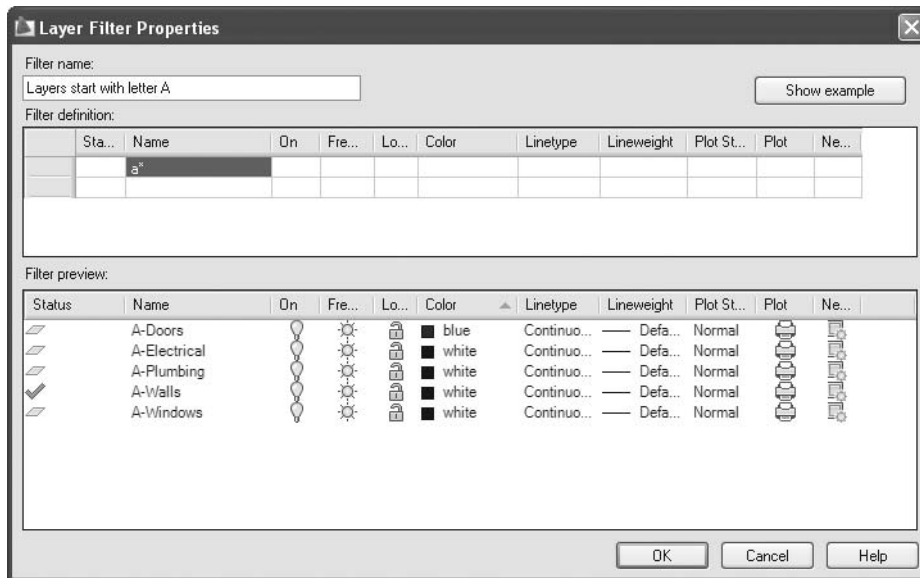
- Right-click the headings of one of the columns. If you do so, you will see:



- Show or hide any column and choose from several available tools.

## 16.6 THE PROPERTY FILTER

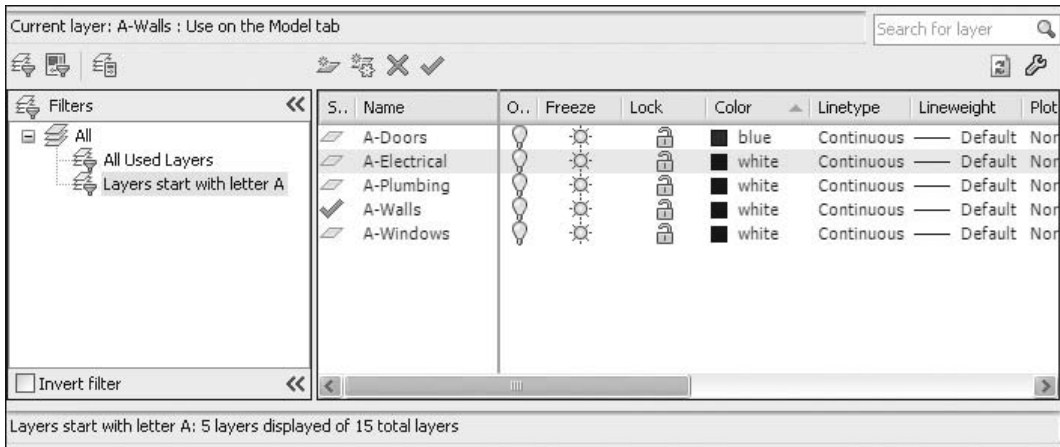
- Complex drawings mean that there will be a large number of layers.
- If you have many layers in the **Layer Properties Manager**, then finding the desired layer to freeze or lock will be difficult.
- AutoCAD provides a great tool to see a selection of layers based on a common property.
- Click the **Property Filter** button. You will see the following dialog box:



- First, fill in a name for the filter.
- If you use one or all of the fields available, you will see an immediate result.
- In the preceding example, the **Name** field has been filled in with **a\***, which indicates that the search should be for any file starting with the letter “a.” The \* is considered to be wild-card, hence it can take the place of anything. You can use it before a letter, after a letter, or both. For example, **\*plum\*** would produce any layer containing the word plum regardless of the beginning, the middle, or the end of the name.
- You can filter using **On/Off**, **Freeze/Thaw**, **Color**, **Linetype**, and so on.



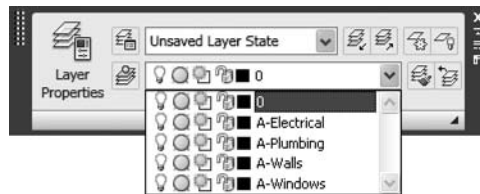
- When you are done, click **OK**. You will see the following:



- In the lower left-hand corner of the palette, you will see:

Layers start with letter A: 5 layers displayed of 15 total layers

- Next, go to the **Home** tab and look at the **Layers** panel to see the following:

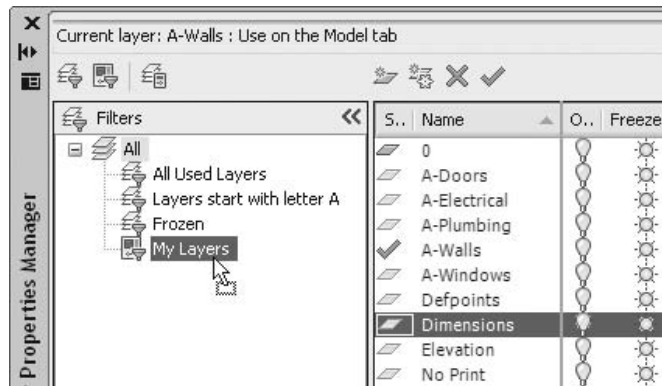


- As you can see, only the layers of the example filter—plus layer 0, which is the current layer in this case—are shown.

## 16.7 THE GROUP FILTER

- We have just seen how the **Property Filter** works. What if you want to create a group of layers and these layers have nothing in common? In this case, you need to create a **Group Filter**.
- Click the **Group Filter** button in the **Filter** pane. You will now have the ability to type in the name of the new **Group Filter**.

- You can now drag and drop layers from the layer list into the filter you have just created.
- See the following illustration:

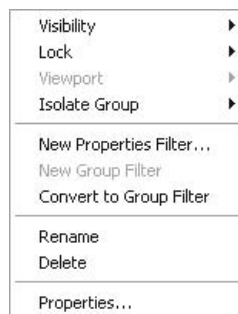


## 16.8 RIGHT-CLICKING THE FILTER

- What will happen if you right-click a **Property Filter** or a **Group Filter**?
- How do filters help speed up production?

### Right-Click a Property Filter

- Since these layers are combined, we can perform some collective actions that will affect all of them at once.
- If you right-click a **Property Filter**, you will get the following menu:



- **Visibility** presents you with a submenu:



- You can turn all layers **On/Off** or you can **Thaw/Freeze** all layers.

- **Lock** presents you with a submenu:



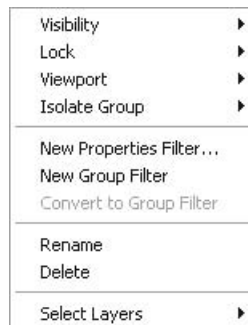
- You can **Lock/Unlock** all layers.
- **Viewport** presents you with a submenu:



- You can **Freeze/Thaw** all of these layers in the current **Viewport**.
- **Isolate Group** will be discussed later in this chapter.
- **New Properties Filter** allows you to create a new **Property Filter**.
- **Convert to Group Filter** will not add a new layer to the list even if it fits the criteria of this filter.
- **Rename/Delete/Properties** allows you to rename or delete a filter, and provides you with **Properties** to modify the filter (another method is to double-click the filter name).

#### Right-Click a Group Filter

- A similar menu will appear if you right-click a **Group Filter**.



- This menu is identical to the **Property Filter** menu, except that the last choice is **Select Layers**.

- This option is useful when you wish to include new layers inside the filter without naming them. Instead, you can select objects that reside in the filter.
- The following prompt will appear:

Add layers of selected objects to filter....:

- Select an object in the desired filter to be added into the group filter.

## THE PROPERTY AND GROUP FILTERS (GENERIC)

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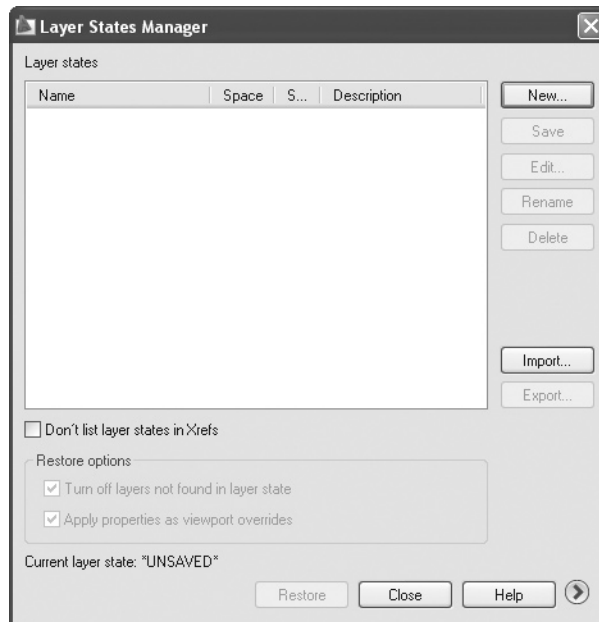


### Exercise 88

- Start AutoCAD 2010.
- Open the file *Exercise\_88.dwg*.
- Start the **Layer Properties Manager**.
- How many layers are in this drawing?
- How many used layers are in this drawing?
- Using the drag-and-drop method, move the **Freeze** column to be the second column at the left.
- Hide the column **Description**.
- Sort layers according to color in descending order. What is the name of the first layer now?
- Create a new **Property Filter** and name it **Structural Layers**. This filter will include all layers that start with “S.”
- How many structural layers are there?
- Create a new **Group Filter** and name it **Annotation Layers**. Include the following layers:
  - **A-Dim**
  - **Text**
- Right-click the **Annotation Layers** filter and choose **Select Layers/Add**. Now, select one of the grid lines and one of the circles.
- Go to the **Annotation Layers** filter. You will find that two more layers were added.
- Right-click the **Annotation Layers** filter and select **Visibility/Frozen**.
- Close the **Layer Properties Manager** and see the effect of these changes.
- Save the file and close it.

## 16.9 THE LAYER STATES MANAGER COMMAND

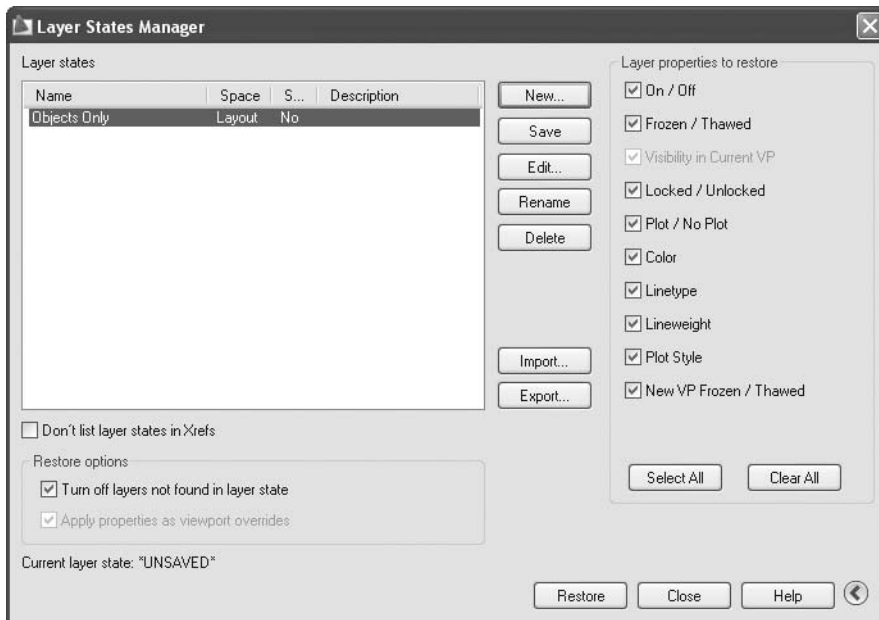
- While you are working on a drawing, you will **Freeze/Thaw**, turn **On/Off**, change colors of layers, and so on.
- At a certain stage of the development process, you can save the current state of the layers as is and retrieve it later.
- From the **Layer Properties Manager**, click the **Layer States Manager** button. You will see the following dialog box:



- Click the **New** button to create a new layer state. The following dialog box will appear:

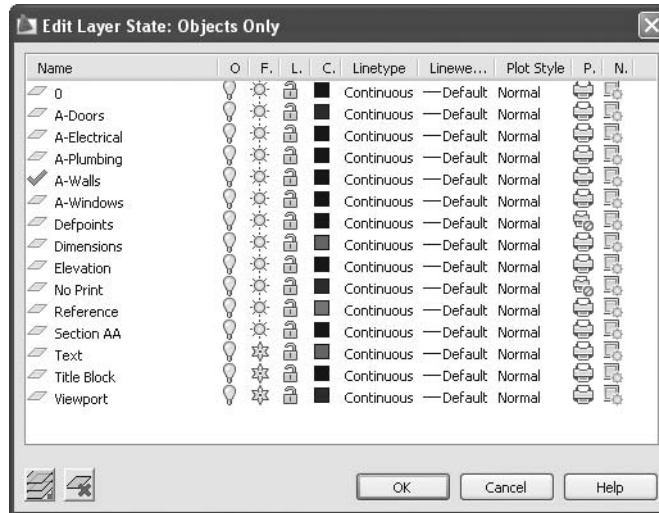


- Type in the name of the new layer state and click **OK**. AutoCAD will save the current state of the layers.
- The question is: Will AutoCAD save everything related to layers or will it save only some of them?
- To answer this question, take a look at the lower right-hand corner of the dialog box. You will see a small round button with an arrow pointing to the right. Click it and you will see the following:



- Under the title **Layer properties to restore**, select the desired properties.

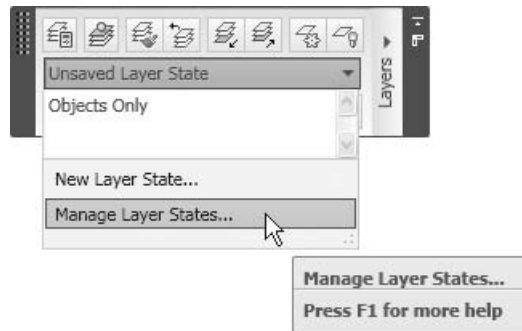
- At any time, you can select buttons such as:
  - **Save** allows you to save the selected layer state.
  - **Edit** allows you to edit the selected layer state. You will see the following dialog box:



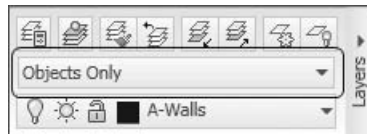
- Change the properties as you wish.
- **Rename/Delete** allows you to rename or delete the selected layer state.
- **Import/Export** allows you to import or export a layer state from other files or to other files, respectively.
- Once you are done, click either the **Restore** button to restore the selected layer states, or the **Close** button to close the dialog box.



- You can issue the same command from outside the **Layer Properties Manager** if you are on the **Home** tab on the **Ribbon**. Using the **Layers** panel, click the pop-up list just above the layers list. Select **Manage Layer States**:



- You can also see the current layer state from the same panel:



## THE LAYER STATES MANAGER (GENERIC)



### Exercise 89

- Start AutoCAD 2010.
- Open the file *Exercise\_89.dwg*.
- Start the **Layer Properties Manager**.
- Start the **Layer States Manager** and create a new state called **All Objects**.
- Save and restore the state.
- Freeze the following layers:
  - **A-Dim**
  - **S-Grid**
  - **S-Grid-Iden**
  - **Text**
- Start the **Layer States Manager** and create a new state called **Objects Only**.
- Make sure all options are selected.
- Save and restore the state.
- Restore the **All Objects** state.

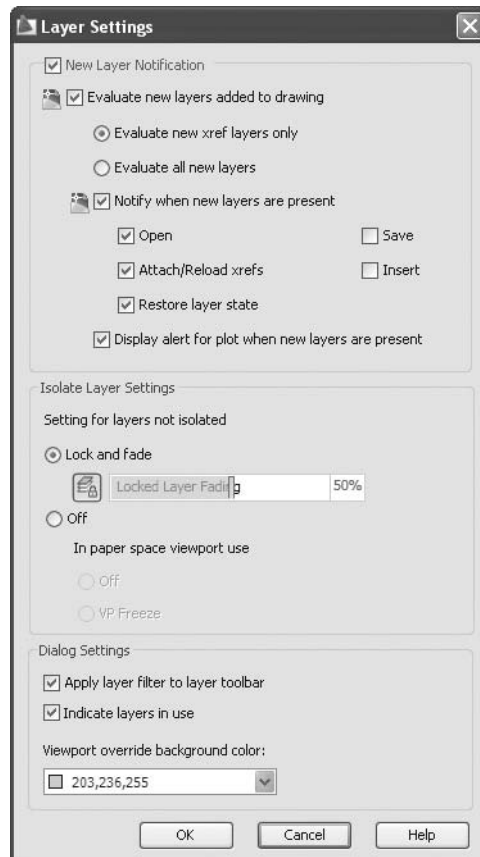


- Select all layers, and then deselect layers **A-Wall** and **A-Wall-Prht**. Freeze all selected layers.
- Start the **Layer States Manager**, create a new state, and call it **Walls Only**.
- Save and restore the state.
- Try the three states you just created.
- Save the file and close it.

## 16.10 THE SETTINGS DIALOG BOX



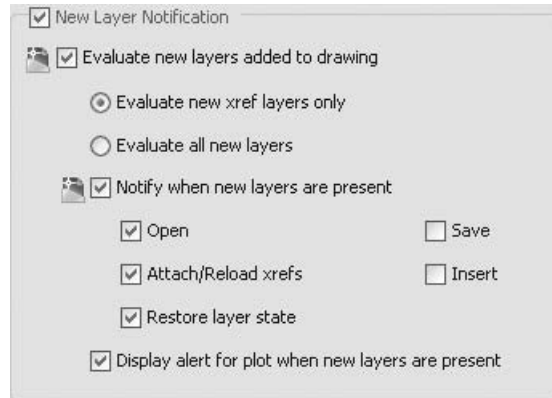
- In the **Layer Properties** dialog box, there is a button called **Settings**. Click it to see the following dialog box:



- This dialog box consists of three parts:
  - **New Layer Notification**
  - **Isolate Layer Settings**
  - **Dialog Settings**

### New Layer Notification

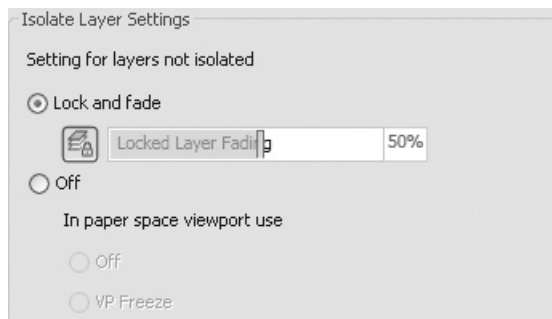
- In accordance with the CAD standards discussed earlier in this chapter, the user shouldn't add any new layers to his or her drawing, whether it's intentional or unintentional.
- AutoCAD will notify the user if new layers are added. See the following:



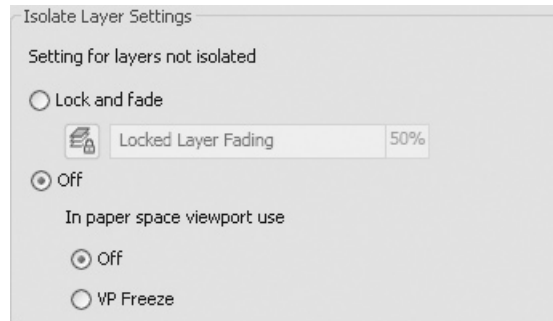
- First, select whether or not you would like to be notified.
- If yes, select whether or not to **Evaluate new layers added to drawing**.
- If yes, select which type of layers (**XREF** layers or all layers).
- Now select when you like to be notified (i.e., when opening the file, when saving the file).
- Finally, you can choose to be alerted if any new layers have been added when you use the **Plot** command.

### Isolate Layer Settings

- The **Isolate** command will be discussed shortly. You must decide how you want the other layers that are not isolated to appear. You have the choice to **Lock and fade** them (you can select the fading percentage) or turn them **Off**. The following shows the user selecting the **Lock and fade** option.

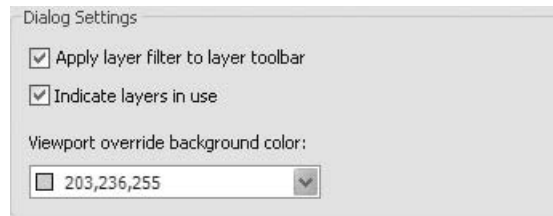


- If you choose the latter, you will see the following view:



### Dialog Settings

- These three settings apply to the **Layer Properties** dialog box. See the following:



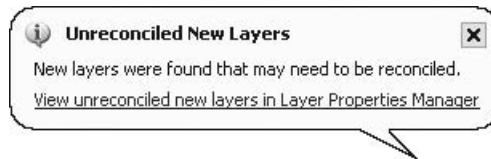
- Select whether to **Apply layer filter to layer toolbar**.
- Select whether to **Indicate layers in use**. See the following:



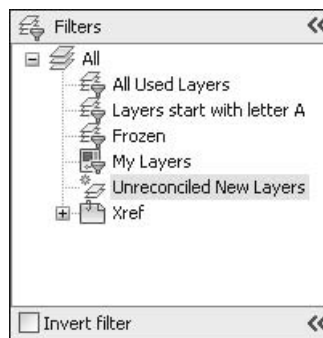
- Layers in use are the layers that contain a single object. Can you tell from the preceding illustration which of these layers are in use, and which are not?
- Earlier, we discussed the layer override in viewports (VP). In order to distinguish them from the others, what is a suitable background color?

### Unreconciled Layers

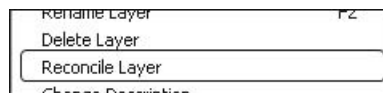
- If a new layer was added, AutoCAD will inform you according to the method you chose in the preceding dialog box.
- An alert message will be displayed in the **Status Bar**:



- When you click **View Unreconciled New Layers** in the **Layer Properties Manager**, you will see something like the following:



- As you can see, AutoCAD created a new filter called **Unreconciled New Layers**. You can see these layers on the right.
- To reconcile any of these layers, right-click the name of the layer and select the **Reconcile Layer** option.



## 16.11 ADVANCED LAYER FUNCTIONS

- We will now cover a group of functions that will make your life much easier!
- Advanced layer functions are functions related to layers that will allow users to carry out a variety of tasks with a single click.

- All of these functions are in the **Home** tab on the **Ribbon** in the **Layers** panel. See the following:



- You will see the **Isolate** and **Unisolate** buttons.
- **Isolate** a layer allows you to turn off (or lock) all other layers except the desired layer. The following prompt will appear:

Current setting: Hide layers, Viewports=Off  
 Select objects on the layer(s) to be isolated or [Settings]:  
**(Select the desired object)**  
 Layer A-Doors has been isolated.

- **Unisolate** is used to turn on (or unlock) the other layers. There are no prompts for this function.



- You will see the **Freeze** and **Off** buttons.
- **Freeze** and **Off** allow you to turn off a layer by selecting an object that resides in the layer. The following prompts will appear:

Current settings: Viewports=Vpfreeze, Block nesting level=Block  
 Select an object on the layer to be frozen or [Settings/Undo]: **(Select the desired object)**  
 Layer "A-Doors" has been frozen.  
 Select an object on the layer to be frozen or [Settings/Undo]: **(Select another object, and so on)**



- You will see the **Turn All Layers On** and **Thaw All Layers** buttons.
- These two commands should be used after using the **Freeze** and/or **Off** buttons in order to reverse their effect.
- There are no prompts for these two commands.



- You will see the **Lock** and **Unlock** buttons.
- **Lock** allows you to lock layers by selecting an object that resides in the desired layer. The following prompt will appear:

Select an object on the layer to be locked: **(Select the desired object)**  
 Layer "A-Plumbing" has been locked.

- You can lock one layer at a time.



- **Unlock** will reverse this command with no prompts.
- You will see the **Change to Current Layer** button.
- **Change the Current Layer** allows you to take selected objects from their layers to the current layer. The following prompts will appear:

Select objects to be changed to the current layer: *(Select an object)*

Select objects to be changed to the current layer: *(Continue selecting, when you are done press [Enter])*

3 objects changed to layer "A-Doors" (the current layer).



- You will see the **Copy Objects to New Layer** button.
- **Copy Objects to New Layer** allows you to create copies of selected objects and transfer them from one layer to another. The following prompts will appear:

Select objects to copy: *(Select desired objects)*

Select objects to copy: *(When you are done press [Enter])*

Select object on destination layer or [Name] <Name>: *(Either type a name of the layer, or select an object in the desired layer)*

3 object(s) copied and placed on layer "Partition".

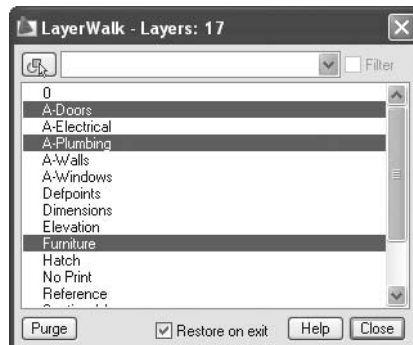
Specify base point or [Displacement/eXit] <eXit>:

Specify second point of displacement or <use first point as displacement>:

- As you can see, the last three prompts are identical to those in the **Copy** command.



- You will see the **Layer Walk** button.
- **Layer Walk** will display a dialog box. By default, all layers are selected. If you click any layer name, the objects that reside in that layer will be shown and the others will disappear.
- You can select multiple layers by holding [Ctrl] and selecting additional layers. See the following example:





- You will see the **Isolate to Current Viewport** button.
- Earlier we learned that you can freeze a layer in one viewport and show it in another.
- Using the **Isolate to Current Viewport** command, you can freeze a layer in all other viewports except the current one. The following prompts will appear:

```
Current settings: Layouts=Current Layout, Block nesting
level=Block
Select an object on the layer to be isolated in viewport or
[Settings/Undo]: (Select the desired object)
Layer Text has been frozen in all viewports but the current
one.
Select an object on the layer to be isolated in viewport or
[Settings/Undo]: (Continue doing the same, when you are done
press [Enter])
```



- You will see the **Merge** button.
- **Merge** allows you to combine one or more layers into another layer by selecting objects that reside in those layers. The last layer selected will remain and the others will be deleted. The following prompts will appear:

```
Select object on layer to merge or [Name]:
Selected layers: A-Doors.
Select object on layer to merge or [Name/Undo]:
Selected layers: A-Doors,A-Windows.
Select object on layer to merge or [Name/Undo]:
Select object on target layer or [Name]:
***** WARNING *****
You are about to merge 2 layers into layer "Partition".
Do you wish to continue? [Yes/No] <No>: Y
Deleting layer "A-Doors".
Deleting layer "A-Windows".
2 layers deleted.
```



- You will see the **Delete** button.
- **Delete** allows you to remove objects in the selected layer and purge it. The following prompts will appear:

```
Select object on layer to delete or [Name]:
Selected layers: Partition.
Select object on layer to delete or [Name/Undo]:
***** WARNING *****
You are about to delete layer "Partition" from this drawing.
```

```
Do you wish to continue? [Yes/No] <No>: Y
Deleting layer "Partition".
1 layer deleted.
```

## SETTINGS DIALOG BOX AND ADVANCED FUNCTIONS (GENERIC)

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### Exercise 90

- Start AutoCAD 2010.
- Open the file *Exercise\_90.dwg*.
- Start the **Layer Properties Manager**. Click the **Settings** button.
- Turn **New Layer Notification** on and make the following changes:
  - **Evaluate new layers added to drawing** = on
  - **Evaluate all new layers** = on
  - **Notify when new layers are present** = on
  - **Open** = on
  - **Save** = on
- In the **Isolate Layer** settings, make the following changes:
  - **Lock and fade** = on
  - **Fade %** = 75%
- In the **Dialog** settings, make the following changes:
  - **Indicate layers in use** = off
- Close the dialog box and the **Layer Properties Manager**.
- Start the **Isolate** command and select one object in the **A-Wall** layer. Press [Enter] to finish the command.
- If you approach any other object, you will notice that the lock symbol appears, which indicates the object is locked. You will also see the fading percentage.
- Use the **Unisolate** command to return to the original file.
- Start the **Layer Walk** command and click inside the dialog box to remove any selection. Select layers **A-Wall**, **A-Wall-Prht**, **A-Door**, and **A-Window**.
- Turn off **Restore on exit** and click **Close**. At the message alert, click **Continue**.
- Start the **Layer Properties Manager**. Notice that the other layers are turned off.
- Close the **Layer Properties Manager**.
- Click the **Turn All Layers On** button.
- Start the **Merge** command, click any object in the layer **A-Wall-Prht** (the inner partition objects), and then press [Enter]. Now select any object on **A-Wall**. When the prompt “Do you wish to continue” appears, click **Yes**. Check the layers. What happened to layer **A-Wall-Prht**?



- Add a new layer and call it **Partition**. Save the file. Did a warning message appear concerning unreconciled layers? If so, click the bubble to check the unreconciled layers. Select the name of the layer, right-click, and select the **Reconcile** option. What happened to the filter?
- Save the file and close it.

## CHAPTER REVIEW

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1. The **Layer Translator** command is a good alternative to CAD standards.
  - a. True
  - b. False
2. If the layers you want to put together have nothing in common, then use the \_\_\_\_\_.
3. You can control whether isolated layers can be turned off or locked.
  - a. True
  - b. False
4. Which one of the following statements is incorrect?
  - a. You have to create a DWS file to be eligible to check **CAD Standards**.
  - b. A DWS is a file you create using the **Save As** command.
  - c. A DWF is a file you create using the **CAD Standards** panel and the **DWS Create** button.
  - d. You have to link your DWG files to DWS files in order to check **CAD Standards**.
5. Which one of these statements about the **Isolate** command is false?
  - a. There is an **Unisolate** command.
  - b. You can isolate more than one layer using the same command.
  - c. Other layers will be frozen.
  - d. You can control the fading percentage from the **Settings** dialog box and the **Layers** panel.
6. If you want to delete a layer and place all of its objects in another layer, you have to use the \_\_\_\_\_ command.

**CHAPTER REVIEW ANSWERS**

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1. b
2. **Group filter**
3. a
4. c
5. c
6. **Merge**

# Chapter 17

# AUTODESK® DESIGN REVIEW AND MARKUP SET MANAGER

## In This Chapter

- ◇ Introduction
- ◇ A Review of the **Publish** Command
- ◇ Autodesk Design Review: An Introduction
- ◇ Two-Dimensional View
- ◇ The **Format** Panel
- ◇ The **Callouts** Panel
- ◇ The **Draw** Panel
- ◇ The **Measure** Panel
- ◇ The **Tab Stamps and Symbols** Panel
- ◇ Editing Markup Objects
- ◇ The **Markups** Palette
- ◇ The **Markup Set Manager**
- ◇ Receiving the DWF Again: Making Comparisons

## 17.1 INTRODUCTION

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- We have already discussed how important a DWF file is to the AutoCAD® user and how it can solve multiple problems.
- In order to view a DWF file, you must have Autodesk Design Review.
- Autodesk Design Review comes with many markup tools that help design teams conduct their review process electronically.
- The following is an exaggerated scenario about how engineering firms did their work prior to electronic reviews:
  - When an engineer (or draftsman) finished his or her drawing, he or she printed it, and then took the printout to the supervisor for review.
  - If the drawing was not lost among the stacks of papers in the office, the supervisor would start adding some revision clouds, remarks, comments, and so on.

- The supervisor would call the engineer (or draftsman) to discuss all the remarks and notes added to the drawing.
- The engineer (or draftsman) would return to his or her desk, load the AutoCAD drawing, fix the mistakes, and make the alterations requested.
- The engineer (or draftsman) would print the drawing and send it again, possibly forgetting to attach the old drawing with the remarks to the new printout.
- The supervisor would take a look at the new drawing and ask for additional adjustments.
- Rarely would there be a record of all the changes on a drawing, which would result in an incorrect final design.
- Reviewing this process, we can easily highlight the cost of:
  - The supplies required to produce the repeated number of drawings, such as ink, paper, and electricity.
  - The time lost when the employee left his or her desk to send the drawing, pick it up again, resend it, and so on. Bearing in mind that he or she might have needed to leave the building if the supervisor was located in another place.
  - The inability to precisely track the changes that took place in the drawing.
  - These issues might have eventually led to errors in the design, hence bad output or product.
- If we truly learn how to use the cycle between Autodesk Design Review and AutoCAD, and we master the tools available, we can conduct an efficient and accurate review process.

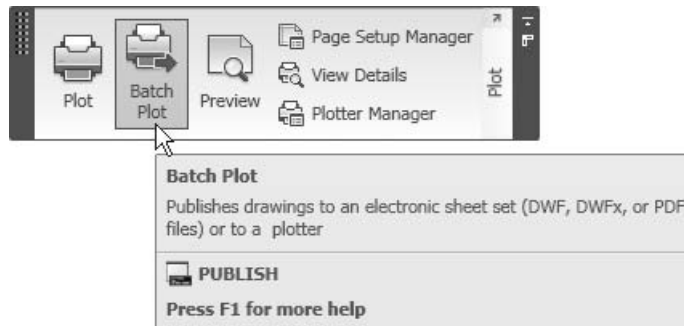
---

## 17.2 A REVIEW OF THE PUBLISH COMMAND

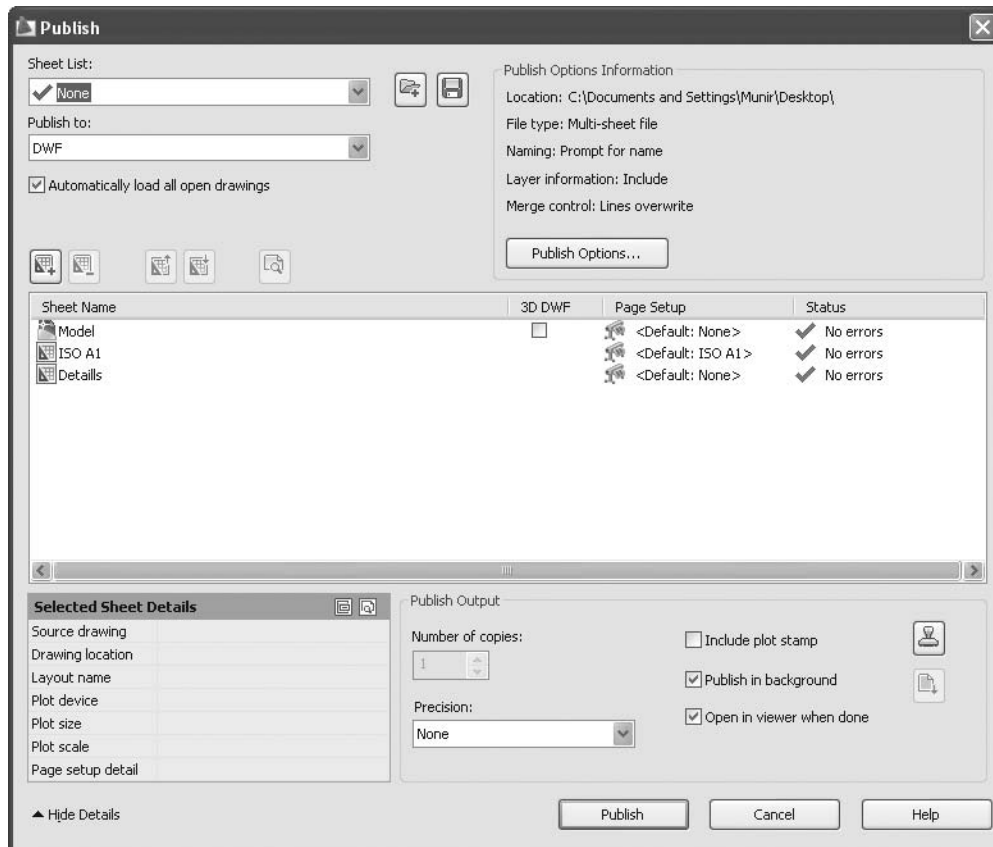
---

- You can publish DWF files in AutoCAD using two different commands:
  - **Plot** command, which uses a *DWF6 ePlot.pc3* printer that will produce a single sheet DWF.
  - **Publish** command, which will produce multisheet DWF file and include more information inside the DWF file such as layers, block information, views, and so on.
- We will concentrate on the **Publish** command.

- Make sure you are at the **Output** tab on the **Ribbon**, and using the **Plot** panel, click the **Batch Plot** button.



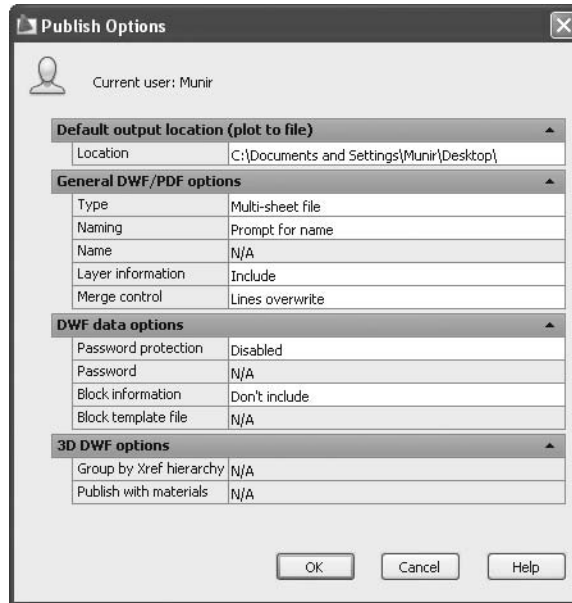
- The following dialog box will appear:



- The dialog box has been covered earlier, but now we want to include some features that have not yet been discussed.

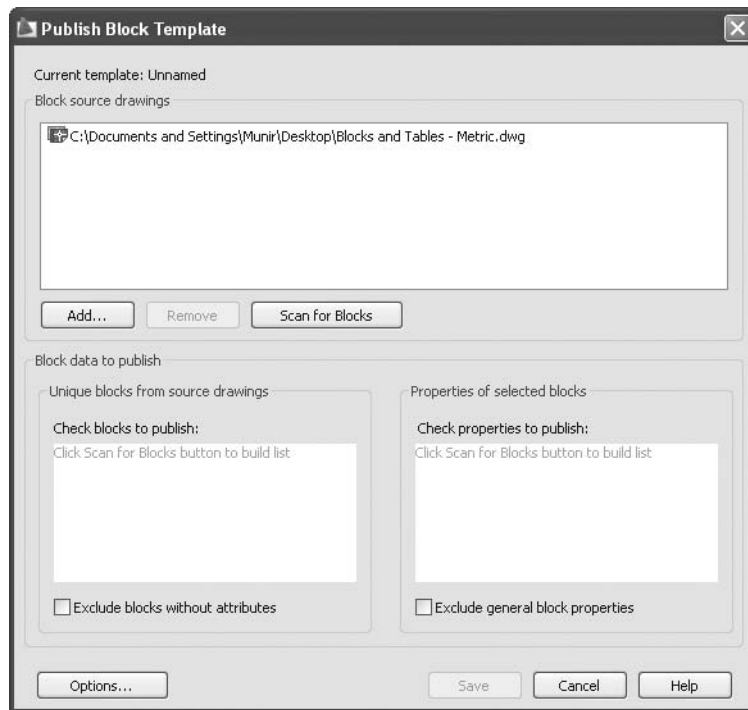
### Publish Options

- The **Publish Options** dialog box dictates what the DWF does and does not contain.
- In this dialog box, you will see the following:

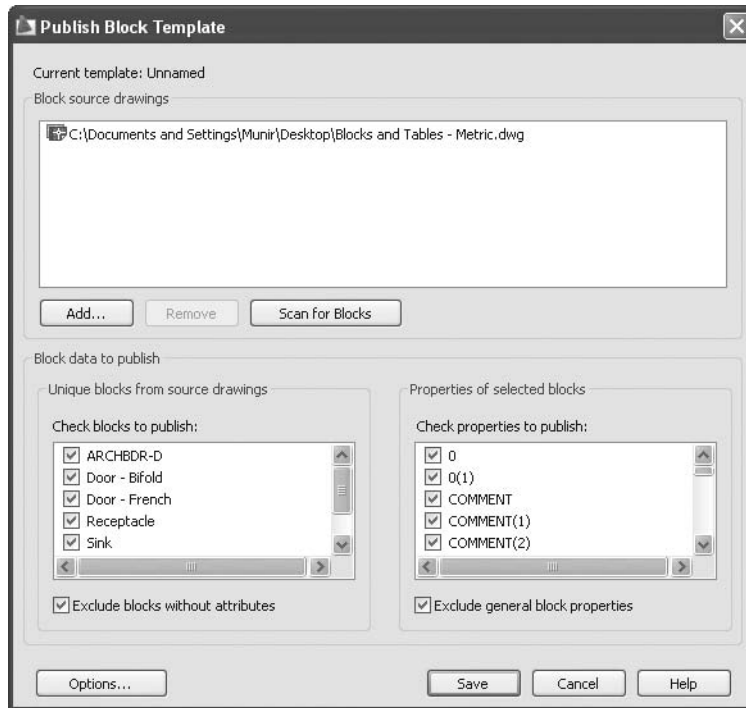


- Do not forget to include the **Layer Information** in the DWF file.
- You can also include the **Block Information** (attributes). You must select the blocks and their attributes to be included in the DWF file.

- Go to the **Block template file**, click on the drop-down menu, and select the **Create** option. The following dialog box will appear:



- Click the **Scan for Blocks** button. You will see the following dialog box:



- AutoCAD will scan all blocks in the drawing (with or without attributes), and then show all blocks and all properties (attributes and non-attributes).
- Select whether to **Exclude blocks without attributes**.
- Select whether to **Exclude general block properties**.
- Once you are done, click **Save**. AutoCAD will show the **Save As** dialog box, and you will save as a DXE file. The next time you want to publish a DWF and you want to include the same information, you can recall this file instead of repeating the entire process.

## A REVIEW OF THE PUBLISH COMMAND (GENERIC)



### Exercise 91

- Start AutoCAD 2010.
- Open the file *Exercise\_91.dwg*.
- Start the **Publish** command.
- Remove the **Model** sheet from the sheet list.
- Make sure that **Publish to** = DWF.



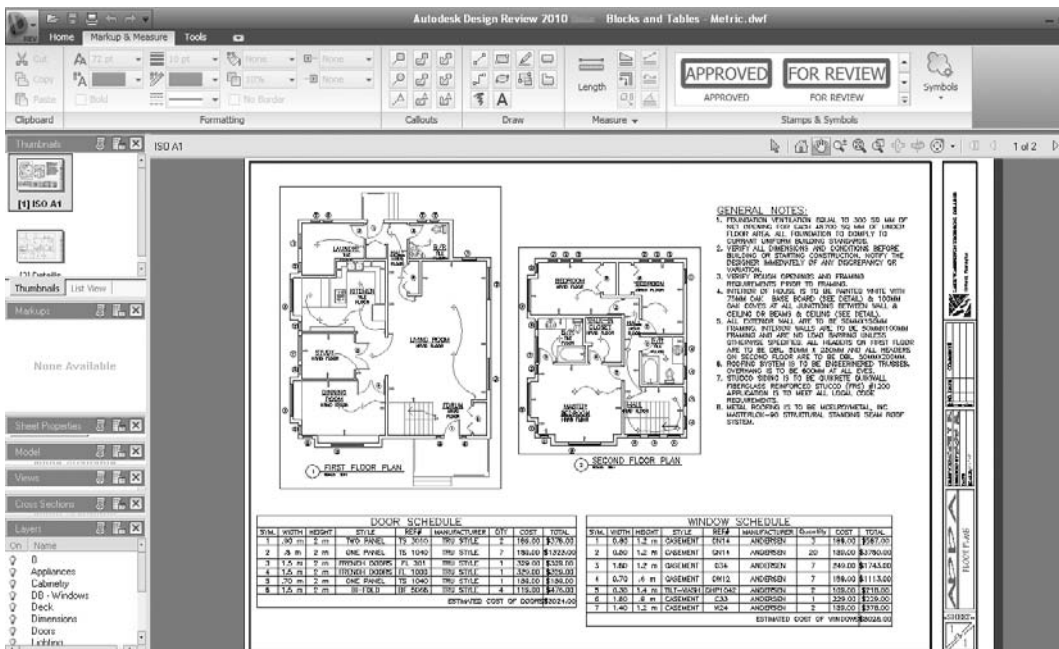
- Click the **Publish Options** button. Be sure of the following:
  - Type** = Multisheet file
  - Layer Information** = Include
  - Change **Precision** = For Architecture
- Save the DWF file in your exercise folder and name it *Exercise\_92.dwf*.
- Save the DWG file and close it.

## 17.3 AUTODESK DESIGN REVIEW: AN INTRODUCTION

- Autodesk Design Review comes in the same DVD as AutoCAD 2010. You can also download it from the Autodesk website for free.
- Double-click the Autodesk Design Review icon on the desktop to start the software (you can also double-click any DWF file and this will start the software as well).



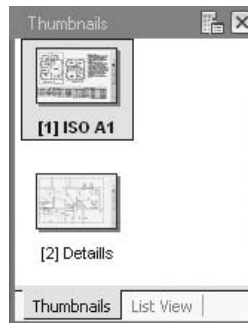
- You will see the following:



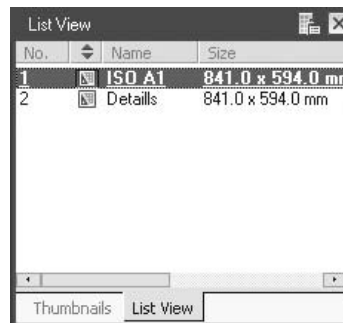
- The interface is similar to the AutoCAD interface (**Ribbons**, **Application Menu**, **Quick Access Toolbar**, and so on).
- There are two other parts on the screen:
  - You will see several palettes on the left-hand side.
  - The remaining portion of the screen is devoted to the **Canvas** and the **Canvas toolbar**.
- By default, there are many palettes open. You can close the ones that you don't need. Also, you can extend one palette over the others by dragging the top edge upward or downward.
- By default, when you open a DWF file, the **Thumbnails**, **Sheet Properties**, **Views**, **Layers**, and **Object Properties** palettes will display some information.

### Thumbnails

- The **Thumbnails** palette will look similar to the following:



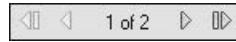
- Another view is the **List View**:



- You can see the sheets by double-clicking a thumbnail or a name in the list.



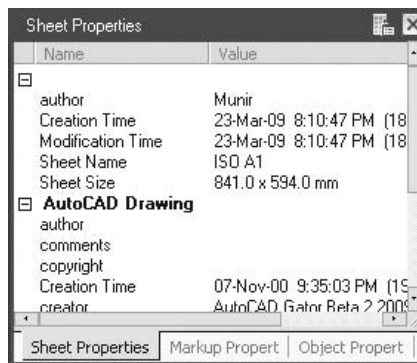
- If we use the **Canvas toolbar** at the top right-hand portion of the **Canvas**, you will see something like the following:



- You can browse the sheets forward and backward. You can also go to the beginning and end of the list of sheets.

### Sheet Properties

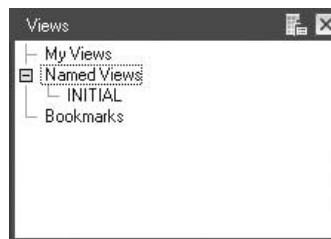
- You will see something like the following:



- **Sheet Properties** list information about the sheet, the AutoCAD drawing, and the sheet set (if it was part of sheet set publishing).

### Views

- The **Views** palette will have information only if you saved views in the drawing file.
- Also, you have to be in the right sheet in order to see the views in the sheet. You will see something like the following:



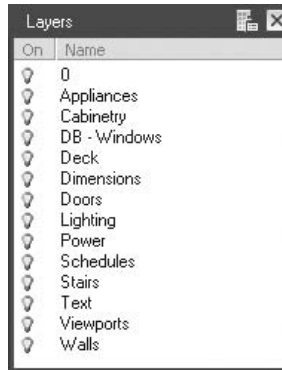
- Simply click the name of the view and it will zoom to it.



- There is always a view called **INITIAL** with the full picture of the sheet.
- This feature is helpful if you made any zoom and you want to return to the full picture again.

### Layers

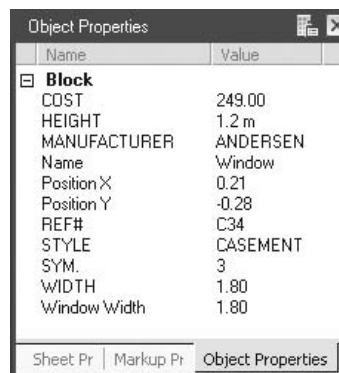
- The **Layers** palette looks something like the following:



- You can turn on or off any layer you wish.

### Object Properties

- If you have block attributes and you choose to include these in your DWF file, then when you click a block that contains an attribute, you will see the following palette:



- You cannot open more than one file in Autodesk Design Review per session.
- Although, you can open more than one session of Autodesk Design Review software at a time.

## 17.4 TWO-DIMENSIONAL VIEW

- You can pan and zoom a DWF file inside Autodesk Design Review using the wheel mouse just as we can with a DWG file in AutoCAD.
- However, there are also some extra functions available in the toolbars and menus that will help you pan and zoom.
- You will see the following:



### Panning and Zooming

- For example, you can use the following:
  - **Zoom Rectangle (Zoom Window)**
  - **Fit to Window**
  - **Zoom In/Out (Zoom Realtime in AutoCAD)**
  - **Pan**
  - **Home**
  - **Select**

### Steering Wheel

- This feature is mainly a three-dimensional tool, but you can use it in two-dimensional drawings as well, but not as effectively. Start the **Steering Wheel** command from the toolbar and you will see the following:

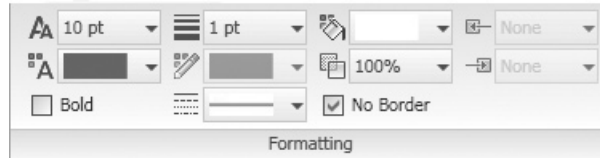


- You can pan if you are at the outer circle. To do so, you click and hold.
- You can zoom in and out if you are at the inner circle. To do so, you click and hold.

## 17.5 THE FORMAT PANEL

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- If at any time you need to add any text or any shape (line, box, circle, and so on) you will need this panel to format your markup.
- The **Format** panel looks like the following:

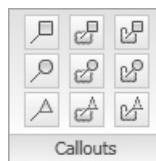


- You can:
  - Turn text bold or regular.
  - Change text size.
  - Change text color.
  - Change object color (line, polyline, rectangle, and so on).
  - Change the fill color of a closed shape.
  - Change the linewidth of an object (only for lines, polylines, rectangle, ellipse, and freehand).

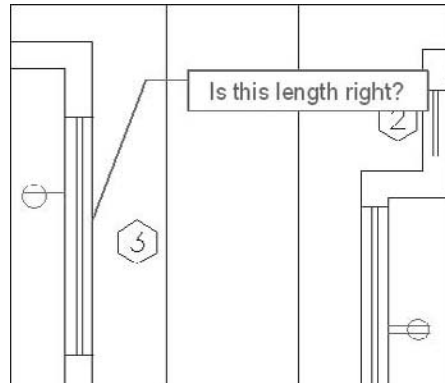
## 17.6 THE CALLOUTS PANEL

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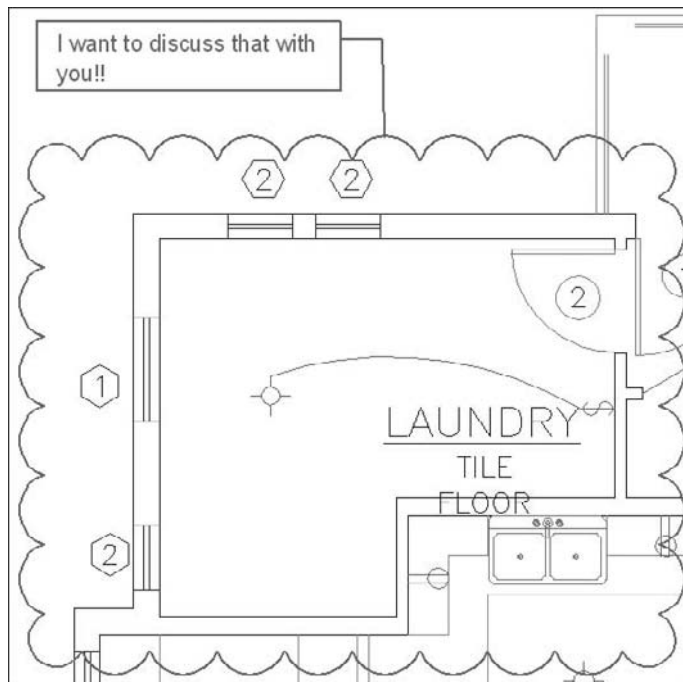
- The **Callouts** panel allows you to place callouts with different shapes.
- The panel looks like the following:



- There are nine different callout shapes, some with revision clouds and some without.
- If you don't have a revision cloud, you will specify two points; the first one at the start of the line segment and the second one at the location of the callout. You will see something like the following:



- If you have a revision cloud, you will first specify the revision cloud by identifying two opposite corners (just like a rectangle), and then you will specify the location of the callout. You will see something like the following:



## 17.7 THE DRAW PANEL

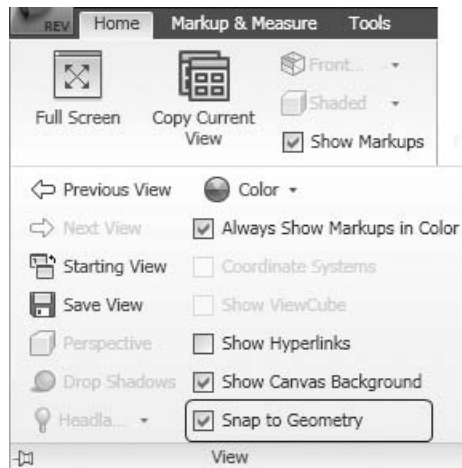
- You can draw any shape you wish in your DWF file.
- The **Draw** panel looks like the following:



- You can draw the following:
  - Line (one segment at a time)
  - Polyline (multisegment)
  - Freehand shape
  - Rectangle
  - Ellipse
  - Write text
  - Freehand Highlighter
  - Rectangle Highlighter
  - Rectangular Cloud
  - Polycloud

## 17.8 THE MEASURE PANEL

- Before you use this tool, make sure you are at the **Home** tab on the **Ribbon**, and using the **View** panel, select the **Snap to Geometry** checkbox. See the following:





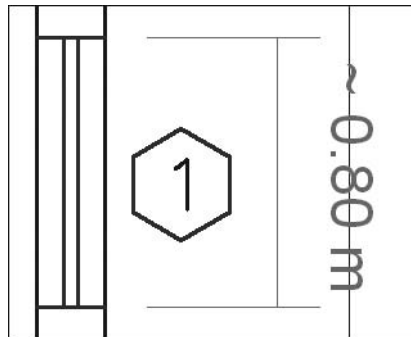
- In the **Measure** panel you see the following:



- Only three options are available for two-dimensional DWF files.

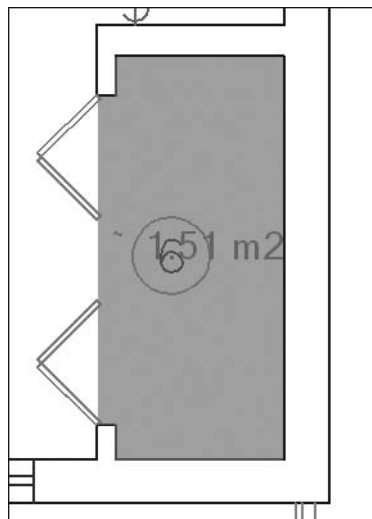
### Length

- You can measure a length between any two points and any angle.
- Specify two points. You will see something like the following:



### Area

- You can measure the area of any shape with straight lines.
- Specify the points. You will see something like that the following:



### Polyline

- You can add up the length of line segments.
- This method allows you to measure the perimeter of an object that consists of straight lines.
- Specify the points and you will see something like the following:



## 17.9 THE TAB STAMPS AND SYMBOLS PANEL

- You can add stamps to the DWF file declaring the current status of the file.
- You will see the following panel:



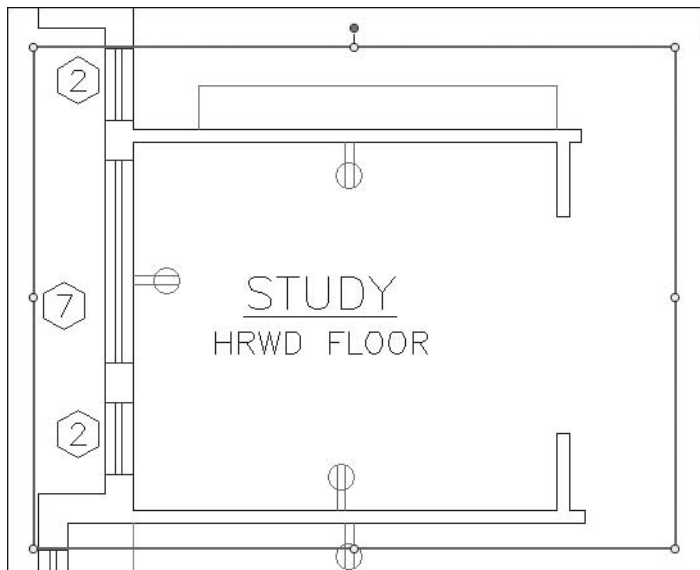
- To do so, you simply click on a suitable place in the file.

## 17.10 EDITING MARKUP OBJECTS

- Once you add **Markup** objects to your DWF file, you will be able to edit these objects.
- In order to edit you must do the following:
  - Make sure you are in the **Select** mode before you begin editing.
  - Click the desired object (you can select multiple **Markup** objects by holding [Shift] and clicking objects).

### Drawing Objects

- **Drawing Objects** can be freehand, line, polyline, rectangle, or ellipse.
- For instance, if you click a rectangle, you will see the following:



- As you can see, the color of the selected object is blue, and eight small circles appear at the edges and midpoints of the lines.
- You can move the rectangle from one place to another by clicking and holding the border of the rectangle.
- You can resize the rectangle using the eight small circles at the edges and the midpoints.
- You can delete it by pressing [Del].

### Callouts

- Click the callout and you will see something like the following:



- You can do the following:
  - Resize the revision cloud.
  - Resize the box containing text.
  - Double-click the text for editing and formatting (you can use the **Format** toolbar to format the text).
  - Delete it by pressing [Del].

### Text Box

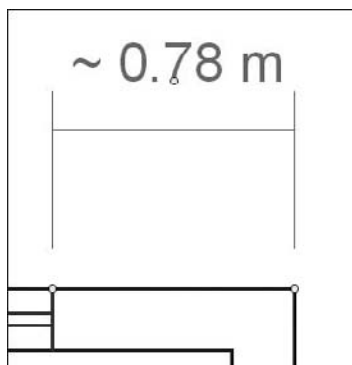
- Click on the text and you will see the following:



- You have the ability to:
  - Resize the text box.
  - Double-click the text for editing and formatting.
  - Delete it by pressing [Del].

**Length (Dimension)**

- Click on the dimension and you will see the following:



- You have the ability to:
  - Change the length to be measured by moving either of the two points.
  - Move the whole dimension block up or down by holding the point near the text.
  - Delete it by pressing [Del].

**Polyline**

- Click on the polyline and you will see the following:



- There are points in each deflection point.
- You have the ability to:
  - Change the shape of the polyline by clicking any point and moving it.
  - Move the whole polyline by clicking and holding.
  - Delete it by pressing [Del].

#### Area

- Click on the area and you will see the following:



- There are points in each deflection point.
- You have the ability to:
  - Change the shape of the measured area by clicking any point and moving it.
  - Move the whole area by clicking and holding.
  - Delete it by pressing [Del].

#### Stamp

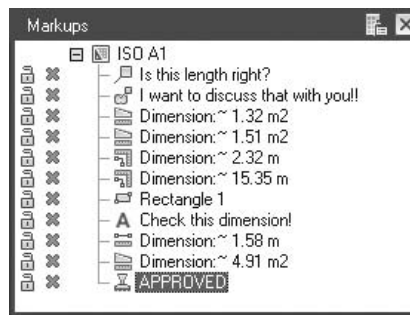
- Click on the area and you will see the following:



- You have the ability to:
  - Resize the stamp.
  - Double-click the text for editing and formatting.
  - Delete it by pressing [Del].

## 17.11 THE MARKUPS PALETTE

- After adding all of your markups, you can view the **Markups** palette, which will look something like the following:



- You will see all the markups sorted by the time of insertion.
- If you click any one of these, it will zoom to it and select it.
- When you are satisfied with your markup process, you need to save all of these markups to your DWF file.
- From the **Application menu**, select **Save**.
- You can also use **Save As** if you would like to save it under another name. However, saving it under the same name is better for revision follow-up and history tracking.

## MARKUP A DWF FILE (GENERIC)



### Exercise 92

- Start AutoCAD 2010.
- Double-click the file you created in the previous exercise, *Exercise\_92.dwf*.
- Browse to see the two sheets. Zoom in, zoom out, and use the pan.
- Using the **Overall** sheet, measure the inner width and height of the **Living room**.
- Calculate the area of the **Study room**. The area is \_\_\_\_\_ m<sup>2</sup>.

- Input a rectangular cloud with a rectangular callout around the small toilet beside the kitchen. In the callout, type the following statement: “I want to see you to discuss the location of the toilet.”
- Turn off the layer **A-Door** and see how many doors turned off.
- Turn off the layer **Dimensions** and see if the other doors disappeared.
- Turn on the layers **A-Door** and **Dimensions**.
- Zoom to the door of the small toilet and make a rectangle callout showing the following statement: “This is the only door in the **A-Door** layer, the rest is in the **Dimensions** layer.”
- Create a text box near the same door and type, “Flip the door opening.”
- Put a stamp in the **Overall** sheet **For Review**.
- Check the **Markups** palette. To see the different markups, you can click on each one to make sure you input the right words and remarks.
- Save the DWF file under the same name and close it.

## 17.12 THE MARKUP SET MANAGER

- After your supervisor has finished his or her remarks on your DWF file, you will get it back to make the requested corrections.
- In order to see the markup in AutoCAD, you must load the **Markup Set Manager**. Make sure you are at the **View** tab on the **Ribbon**, and using the **Palettes** panel, click the **Markup Set Manager**:



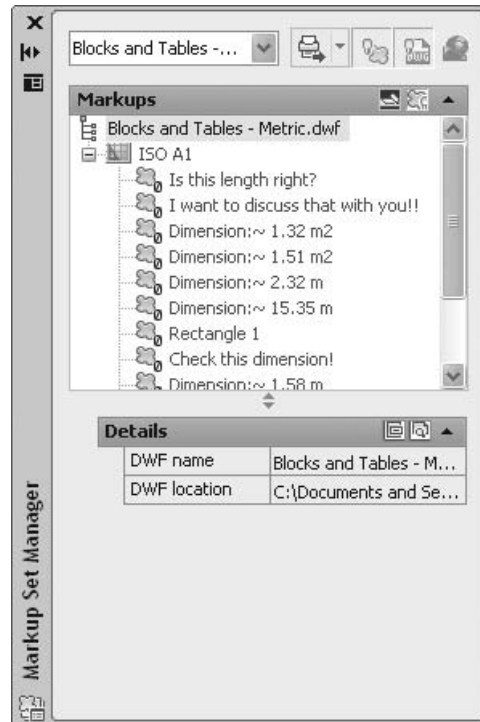


- You will see the following palette:



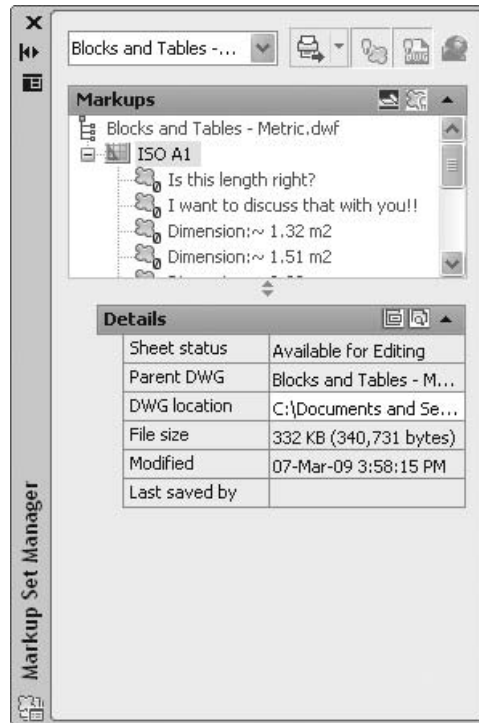
- Select **Open** from the drop-down menu. An **Open file** dialog box will appear, enabling you to select a DWF file with markups.

- You will see the following:



- You can see the markup as a tree.

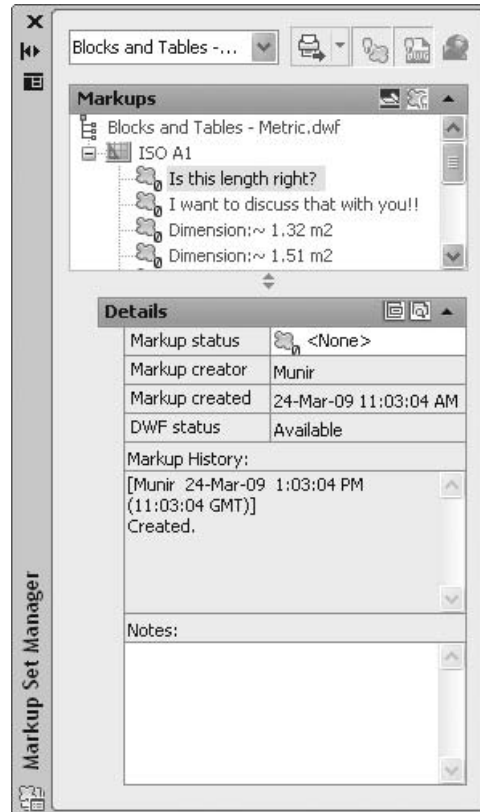
- If you click the sheet node of the tree using the **Details** button, you will see the following:



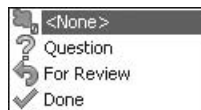
- The two most important pieces of information are **Parent DWG** and **DWG location**. As you can see, the only editable field is the location, which means if your file has moved for one reason or another, you can use the **Markup Set Manager** to find the file again.
- You can see the markup as a tree. If you click (not double-click) one of the markups, you can see **Details** and **Preview**.

### Details

- You will see the following:



- There are six things you must take care of. We will now discuss these items.
  - **Markup status** allows you to set the status of your DWF. You have the following choices:

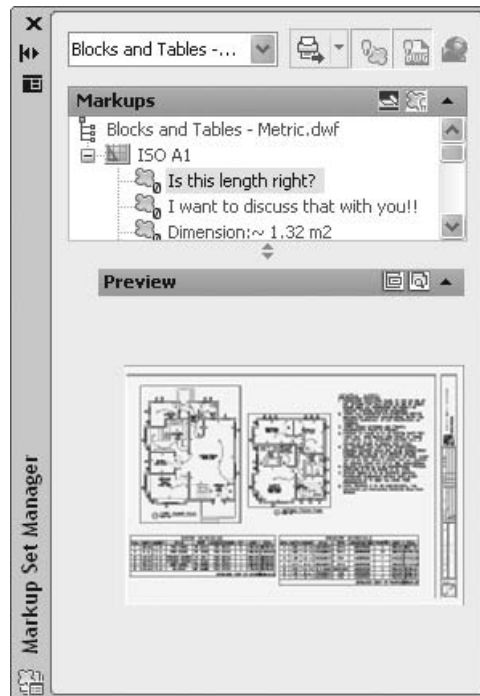


- You can post the status according to your case. For example, if you did what your supervisor requested, then change the status to **Done**.
- **Markup creator**, **Markup created** allows you to add the name of the person who made the markups and the time the markups were made.

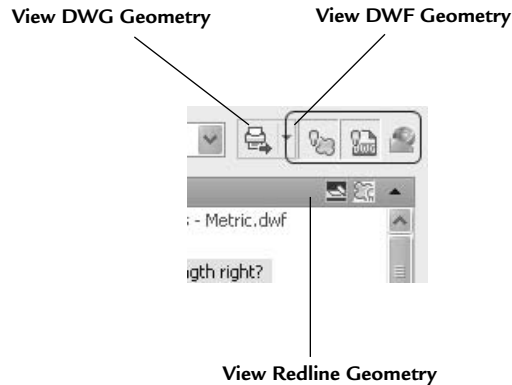
- **DWF status** tells you whether this DWF is available for editing.
- **Markup History** tracks the markups (who made them and when).
- **Notes** allows you to post special notes.

### Preview

- You will see the following:



- If you double-click any markup, the original DWG will be opened along with the DWF file containing the markups. AutoCAD will zoom to the markup selected. At the top right-hand side of the palette you will see the following:



- The two buttons on the right cancel each other out; therefore, by default, the first is turned on and the second is turned off.
- When you make your changes, turn off the DWF geometry.
- The third button allows you to turn off the redlining that took place in the Autodesk Design Review software.
- Once you have made your changes, you need to republish the DWF file with the new adjustments. Click the **Republish Markup DWF** button. You will see the following:

Republish All Sheets  
Republish Markup Sheets

- Choose between **Republish All Sheets** or **Republish Markup Sheets** only.
- It is highly recommended to rename to indicate the revision number (e.g., *name\_rev\_01.dwf*). Thus, you can make comparisons.

## MARKUP SET MANAGER (GENERIC)



### Exercise 93

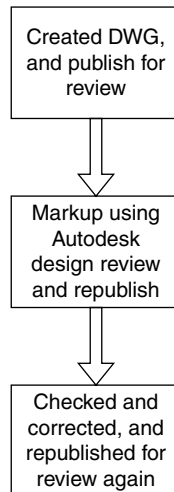
- Start AutoCAD 2010.
- Create a new file.
- Make sure you are at the **View** tab on the **Ribbon**, and using the **Palettes** panel, click the **Markup Set Manager**.
- Click **Open** and locate the DWF file you changed in Exercise 92.
- Double-click **ISO A1 – Overall**. The file will open with markups.
- Double-click **Dimension: ~ 4.75 m and 2.90 m** and change the markup status to **Done**.
- Double-click **Dimension: ~ 14.63 m<sup>2</sup>** and change the markup status to **Done**.

- Double-click “I want to see you to discuss the toilet location” and change the markup status to **Question**. Type “Kindly specify time and location” in the notes section.
- Change the layer of the doors to layer **A-Doors** (you can do so in the **Model** space). Also, flip the door of the small toilet to open to the left rather than the right.
- Double-click “This is the only door ...” and change the markup status to **Done**.
- Change the status of “Flip the door opening” to **Done**.
- Double-click **For Review** and change the markup status to **Done**.
- **Republish Markup Sheets**, name the file *Exercise\_93\_Rev\_01.dwf*, and save it in the exercise folder.
- Close the **Markup** DWF.
- Save and close the DWG file.

### 17.13 RECEIVING THE DWF AGAIN: MAKING COMPARISONS

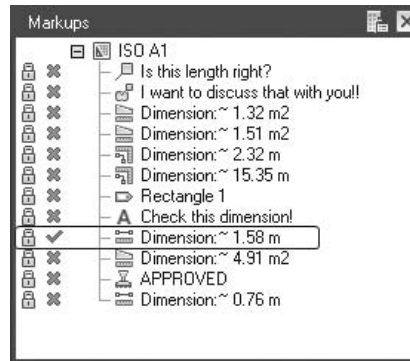
---

- Up until now, we have performed the following steps:

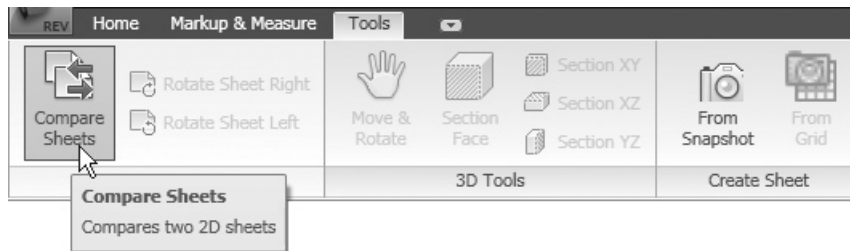


- The next step is for the supervisor to take a look at the DWF again, and compare the first file to the new file to see if the appropriate edits were made.

- When you open the modified file, check the **Markups** palette, and you will see something like the following:



- This palette will give you an idea about the markups and their current status.
- In our example, the user marked the status **Done**.
- To be certain, you can compare the two files.
- To review the finished file, make sure you are at the **Tools** tab on the **Ribbon**, and using the **Canvas** panel, click **Compare Sheets**:

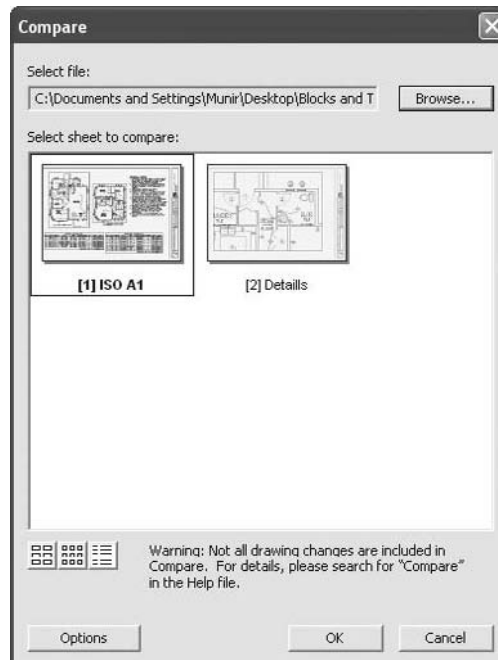




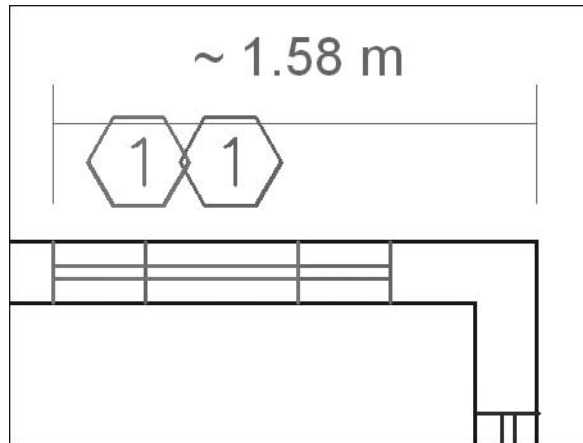
- The following dialog box will appear:



- Click the **Browse** button and select your old file. You will see something like the following:



- Click **OK**.
- You will see something like the following:



- You can see the markup, the old position of the window, and the new position of the window.

## COMPARING TWO DWF FILES (GENERIC)



### Exercise 94

- Start Autodesk Design Review.
- Open the file *Exercise\_93\_Rev\_01.dwf*, which you created in Exercise 93.
- Click the only markup with a question mark. Go to the **Markup Properties** palette.
- You can see the label and its current status. Check the **History** field and you can see the note that you posted in AutoCAD.
- Click **Compare Sheets**, load the old file (*Exercise\_92.dwf*), and then select **ISO A1 – Overall**.
- You can see the previous status of the door and the new status of the new door.
- Save the DWF file and close it.

---

**CHAPTER REVIEW**

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1. You can include the layer information in the DWF file.
  - a. True
  - b. False
2. The name of software that comes with AutoCAD that can read and markup a DWF file is \_\_\_\_\_.
3. You cannot do the following in a DWF file while viewing it:
  - a. Measure a distance.
  - b. Calculate an area.
  - c. Change the layer of an object.
  - d. Add a cloud with callout.
4. Which of these statements is *not* true about marking up?
  - a. You can republish DWF files with modifications in AutoCAD.
  - b. AutoCAD cannot load DWF files.
  - c. You can change the status of the markup inside AutoCAD.
  - d. The **Markup Set Manager** is the tool to load DWF files.
5. You can input a markup with hexagon shaped callout.
  - a. True
  - b. False

---

**CHAPTER REVIEW ANSWERS**

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1. a
2. Autodesk Design Review
3. c
4. b
5. b



# HOW TO CREATE A TEMPLATE FILE

## In This Appendix

- ◇ Introduction
- ◇ Which Elements Are Included in a Template File?
- ◇ How to Create a Template File

### **A.1 INTRODUCTION**

---

- Companies using AutoCAD® are always looking for better ways to:
  - Unify their work to a certain standard (homemade or international).
  - Speed up the process of producing a drawing.
- The answer to these two issues is to create a template file.
- Template files will reassure the decision makers in any company that all the premade settings for the drawings are already done in the templates. This will cut production time by at least 30%.
- Also, templates—which will be provided to all users in the company—will guarantee that these people will be using the same source; accordingly, no personal initiative will be allowed and the results will be uniform.
- Template files are \*.dwt.

### **A.2 WHICH ELEMENTS ARE INCLUDED IN A TEMPLATE FILE?**

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- These are the elements included in a template file:
  - **Drawing units**
  - **Drawing limits**
  - **Grid** and **Snap** settings
  - **Layers**
  - **Linetypes**
  - **Text Styles**
  - **Dimension Styles**
  - **Table Styles**

- **Layouts** (including **Border blocks** and **Viewports**)
- **Page Setups**
- **Plot Style tables**



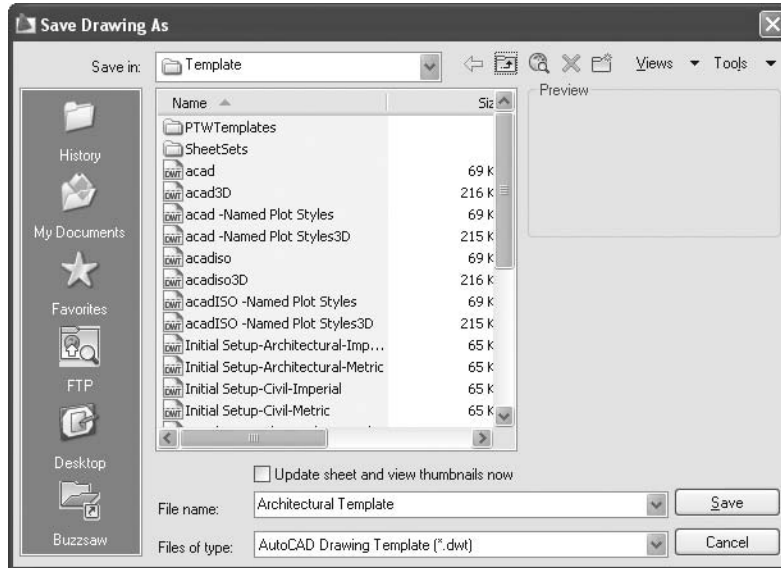
- There is no need to include blocks in the template file. Instead, store them in files with each category in a separate file, such as Architectural, Civil, Mechanical, etc.).
- You cannot save **Tool Palettes** inside template files because **Tool Palettes** will be available for all files in a single computer.

### A.3 HOW TO CREATE A TEMPLATE FILE

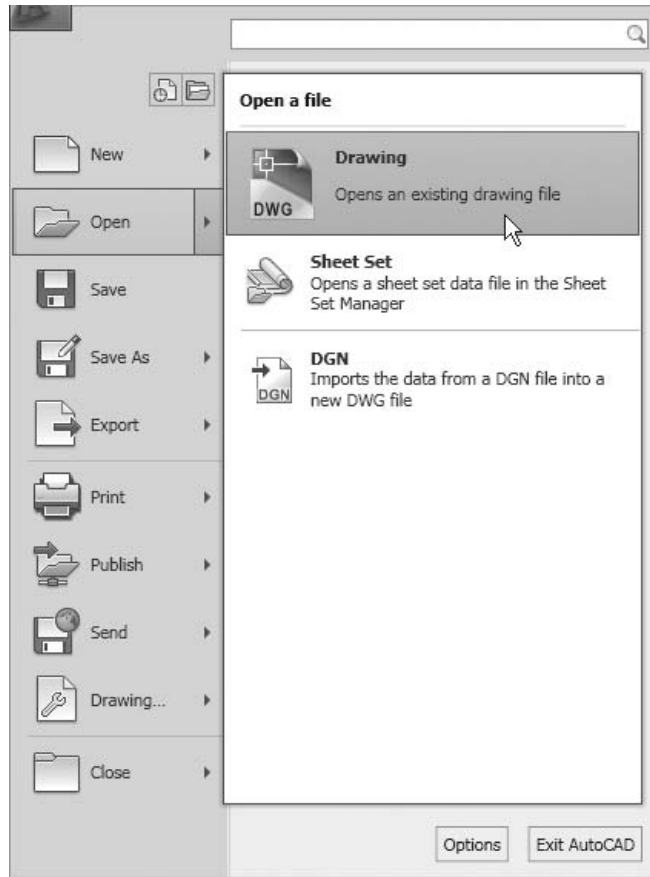
- Start AutoCAD 2010.
- Complete the paperwork to prepare the settings required. This step may involve consultation with other people who operate AutoCAD in the same company.
- Create a new file using the simplest template file, *acad.dwt*.
- The new file will contain the minimum drawing requirements.
- Build all of the necessary elements inside this file.
- Once you are done, select **Save As** and **AutoCAD Drawing Template** from the application menu.



- The following dialog box will appear:

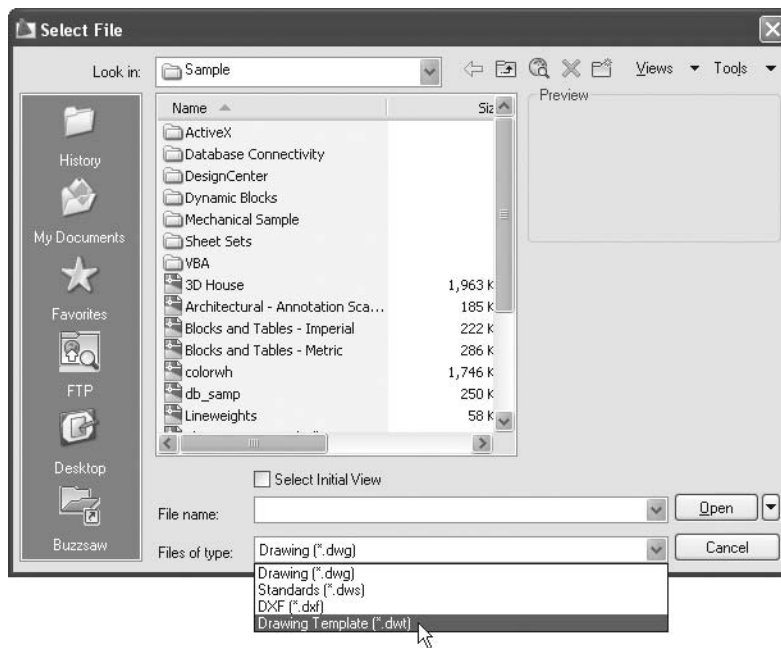


- Type in the template file name.
- By default, you can save your file in the template folder that comes with the software.
- Alternately, you can create a new folder to accommodate all of your template files.
- NOTE ■ It is highly recommended that you store your files in a different folder away from the AutoCAD folders.
- You can create as many templates as you wish.
- If you want to edit an existing template, perform the following tasks:
  - Select **Open/Drawing** from the application menu:



- Select **Drawing Template (\*.dwt)** from **Files of type:**





- Open the desired template and make the desired changes.
- Save it under the same name or use a new name.



**In This Appendix**

- ◇ Introduction
- ◇ The **Distance** Command
- ◇ The **Radius** Command
- ◇ The **Angle** Command
- ◇ The **Area** Command

## **B.1 INTRODUCTION**

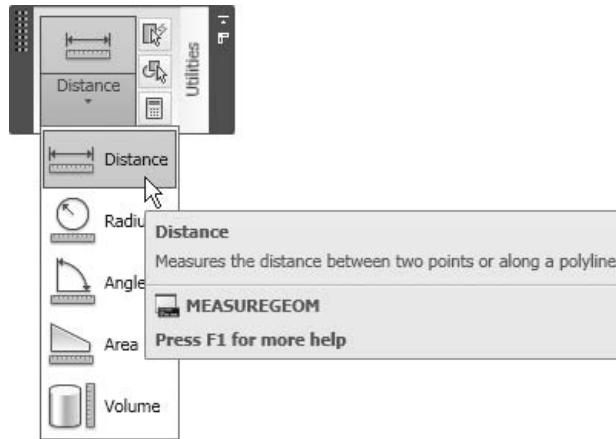
---

- Inquiry commands are used to:
  - Measure a distance between two points.
  - Measure the radius of a circle or an arc.
  - Measure an angle.
  - Calculate the area between points or of an object.
- AutoCAD® provides a single command that includes all of these functions; this command is the **MEASUREGEOM** command.
- To reach this command, make sure you are in the **Home** tab on the **Ribbon**, and select the **Utilities** panel.

## **B.2 THE DISTANCE COMMAND**

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- The **Distance** command is used to measure the distance between two points.
- Make sure you are in the **Home** tab on the **Ribbon**, and, selecting the **Utilities** panel, click the **Distance** button:



- AutoCAD will display the following prompt:

Specify first point: (*Specify first point*)

Specify second point: (*Specify Second point*)

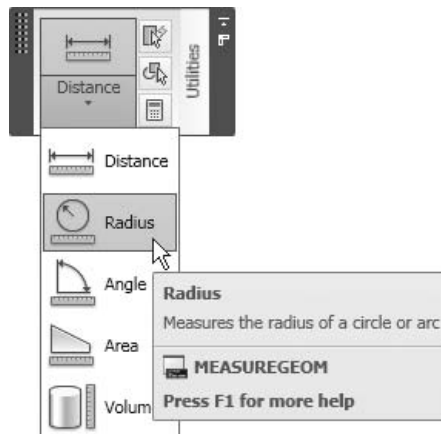
- Click on the two points desired. AutoCAD will display something like:

Distance = 10.0000, Angle in XY Plane = 0, Angle from XY Plane = 0

Delta X = 10.0000, Delta Y = 0.0000, Delta Z = 0.0000

### B.3 THE RADIUS COMMAND

- The **Radius** command is used to measure the radius of a circle or an arc.
- Make sure you are in the **Home** tab on the **Ribbon**, and, selecting the **Utilities** panel, click the **Radius** button:



- AutoCAD will display the following prompt:

Select arc or circle: (*Select the desired arc or circle*)

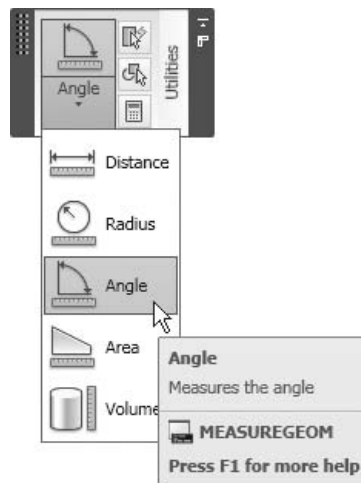
- Select the desired arc or circle. AutoCAD will display something like:

Radius = 6.6294

Diameter = 13.2588

## B.4 THE ANGLE COMMAND

- The **Angle** command is used to measure the angle between two lines included in an arc, between a circle's center and two points, or between a selected vertex and two points.
- Make sure you are in the **Home** tab on the **Ribbon**, and, selecting the **Utilities** panel, click the **Angle** button:



- AutoCAD will display the following prompt:

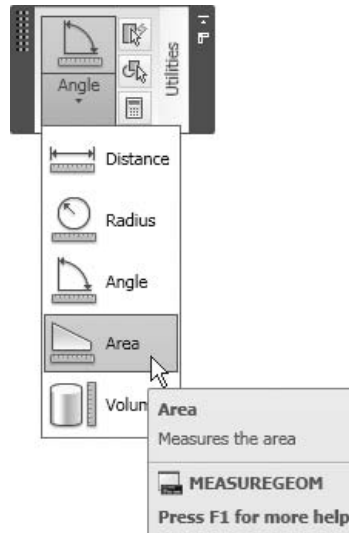
Select arc, circle, line, or <Specify vertex>:

- These are identical to the requirements of the **Angular dimension**.
- Select the desired arc or circle. AutoCAD will display something like:

Angle = 120°

## B.5 THE AREA COMMAND

- The **Area** command is used to calculate the area between points or the area of an object.
- Make sure you are in the **Home** tab on the **Ribbon**, and, selecting the **Utilities** panel, click the **Area** button:



- You can calculate the area for:
  - Points (assuming there are lines connecting them)
  - Objects such as a circle or a polyline (closed or open)
- You can calculate two types of areas:
  - Simple area (single area)
  - Complex area (to determine the net area)
- If you start the **Area** command and specify the points or select the object, AutoCAD will calculate the simple area.
- To calculate the complex area, you must start with either **Add** or **Subtract**.
- Click the **Area** button.
- AutoCAD will display the following prompt:

Specify first corner point or [Object/Add  
area/Subtract area/eXit] <Object>:

### Specify the First Corner

- A simple area consists of points connected by lines or arcs. Specify the first point and AutoCAD will prompt:

Specify next point or [Arc/Length/Undo]:  
Specify next point or [Arc/Length/Undo]:  
Specify next point or [Arc/Length/Undo/Total] <Total>:  
**(After the third point, you can ask for Total value of area measured)**  
Specify next point or [Arc/Length/Undo/Total] <Total>:

- Continue specifying points until you press [Enter] to get the total value of the measured area. The following message will appear:

Area = 33.3750, Perimeter = 23.6264

### Object

- To calculate the area by selecting an object, such as a circle or polyline, press [Enter] as the **Object** option is the default option. You can also type **O** or right-click and select **Object**. AutoCAD will prompt:

Select objects:

- Once you select the desired object, AutoCAD will report the following (in this example the object is a circle):

Area = 28.2743, Circumference = 18.8496

### Add Area/Subtract Area

- You need the **Add area** mode or **Subtract area** mode in order to calculate a complex area—which are areas inside areas—in order to get the net area.
- Start with either one of these two modes and AutoCAD will assume that you are starting with **Area** = 0.00. Hence, you will add the outer area and then subtract the inner areas, or you can subtract the inner areas and then add the outer areas.
- Assume you started with the **Add area** mode. AutoCAD will prompt you to:

Specify first corner point or [Object/Subtract area/eXit]:

- You can specify area(s) using either points or an object. When you are done, switch to **Subtract area** mode, and so on.
- While you are adding and subtracting, AutoCAD will give you the current value of the area until the last area has been added/subtracted.
- Once you are done, press [Enter] twice and AutoCAD will report the final value of the area.

- The following are possible prompts for **Add area/Subtract area**:

```
Specify first corner point or [Object/Add
area/Subtract area/eXit] <Object>: A
Specify first corner point or [Object/Subtract
area/eXit]: O
(ADD mode) Select objects:
```

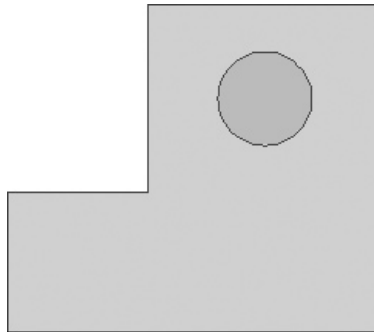
```
Area = 44.0000, Perimeter = 30.0000
Total area = 44.0000
(ADD mode) Select objects:
Area = 44.0000, Perimeter = 30.0000
Total area = 44.0000
```

```
Specify first corner point or [Object/Subtract
area/eXit]: S
Specify first corner point or [Object/Add area/eXit]:
O
(SUBTRACT mode) Select objects:
Area = 3.1416, Circumference = 6.2832
Total area = 40.8584
(SUBTRACT mode) Select objects:
```

```
Specify first corner point or [Object/Add area/eXit]:
```

```
Total area = 40.8584
```

- You will see something like the following illustration:





**In This Chapter**

- ◇ A General Introduction
- ◇ Workspace: An Introduction
- ◇ Workspace: The Commands
- ◇ Ribbons

**C.1 A GENERAL INTRODUCTION**

---

- Each of us likes to change the interface of any software to suit our personal preferences.
- In most of the software packages we call this “interface customization.”
- AutoCAD® provides two tools for interface customization:
  - Workspace
  - Profile
- Starting with AutoCAD 2009, AutoCAD adopted a new interface that looks and feels like Microsoft® Office 2007 using **Ribbons**.
- **Ribbons** consist of two parts:
  - Tabs
  - Panels
- **Ribbons** replace toolbars and give the user more screen space.
- However, if you still like the old AutoCAD interface, you can get it back by restoring other workspaces that come with AutoCAD.

**C.2 WORKSPACE: AN INTRODUCTION**

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- Showing or hiding your choice from the following list is considered a workspace:
  - **Menus**
  - **Toolbars**

- **Palettes**
- **Ribbons (Tabs and Panels)**
- The workspace will memorize the location of each choice and whether it is to be floating or docked.
- In very simple terms, you can decide what to show and what to hide, where to locate each item, and then save it as your workspace.
- As a result, you can restore your settings at any time.

### C.3 WORKSPACE: THE COMMANDS

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- AutoCAD comes with three predefined workspaces:
  - **2D Drafting & Annotation** is the default workspace.
  - **3D Modeling** is used for three-dimensional commands.
  - **AutoCAD Classic** restores the old look before AutoCAD 2009.
- Use the button on the **Status Bar** to reach these three predefined workspaces and other commands:



- When you click it, the following menu will appear:



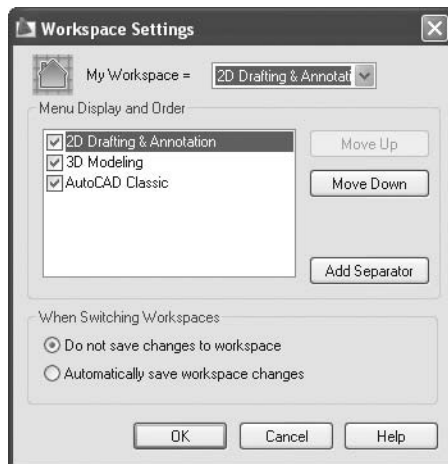
- You can restore any of the three workspaces by selecting the name.

#### Save Current As

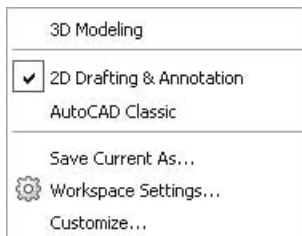
- If you made any changes to your current workspace, you can save it under a new name.

### Workspace Settings

- If you start the **Workspace Settings** command, you will see the following dialog box:

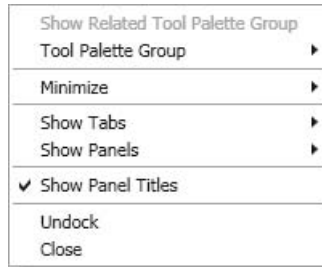


- You can make the following changes:
  - Change the order of the saved workspaces by moving them up or down. The position will affect their listing in the menu on the **Status Bar**.
  - You can add a separator (simple line) between the workspace names in the menu.
  - By default, AutoCAD will not save any changes you make to your workspace unless you save it under another name. However, you can direct AutoCAD to save them automatically.
- The following demonstrates changes made to the menu using the previous commands. We changed the order and added a separator:



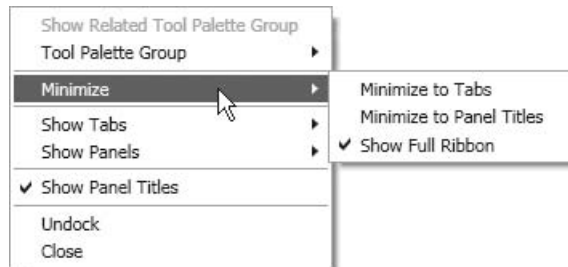
## C.4 RIBBONS

- Since **Ribbons** are an essential component of the new interface, we will devote some time to learning how to deal with them effectively.
- If you right-click any tab you will see the following menu:



### Minimize

- If you click **Minimize** you will see the following menu:



- This will allow you to:
  - **Minimize to Tabs**



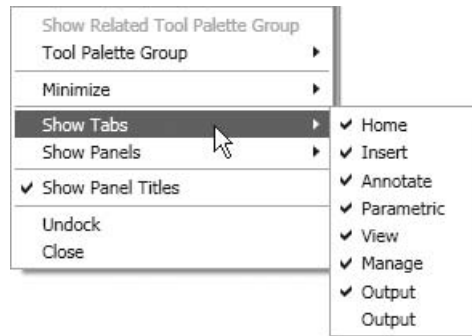
- **Minimize to Panel Titles**



- **Show Full Ribbon**

### Show Tabs

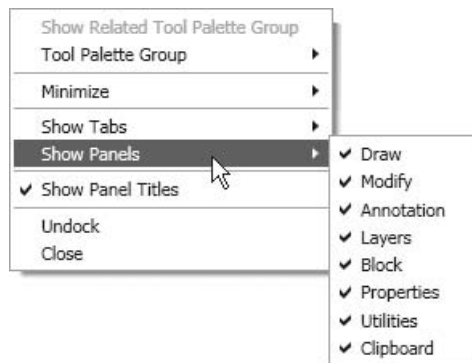
- If you click **Show Tabs**, you will see the following menu:



- You can select which of the existing tabs you want to show or hide.

### Show Panels

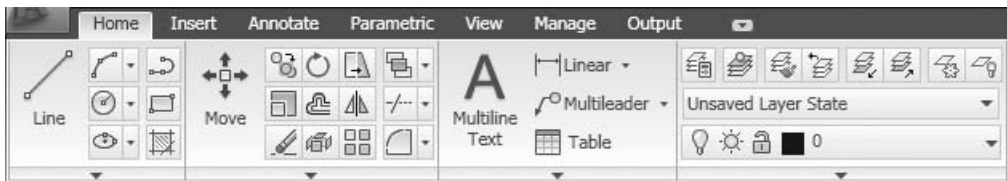
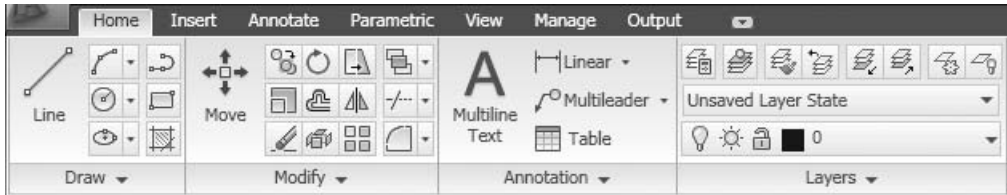
- If you click **Show Panels**, you will see the following menu:



- You can select which of the panels you want to show or hide in a certain tab.

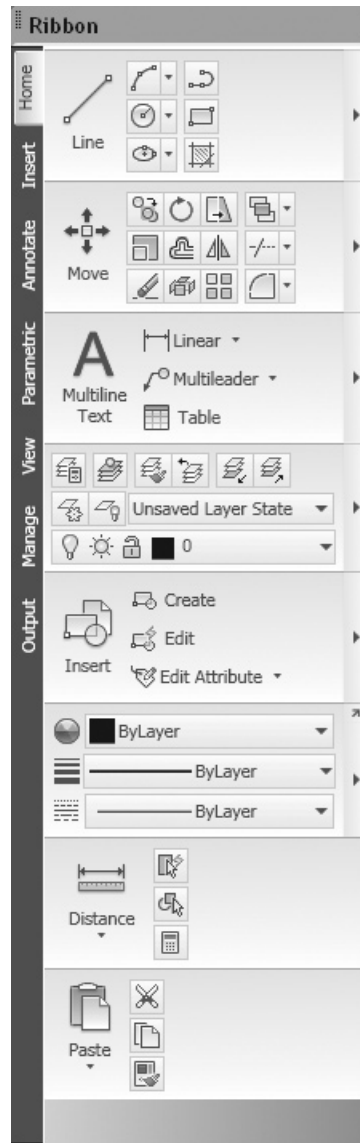
### Show Panel Titles

- Select whether or not to show the panel titles. See the following illustrations:



### Undock

- By default, the **Ribbon** is docked at the top of the screen.
- You can undock it to make it float, or dock it at the right, left, or bottom of the screen.
- The following is an example of a **Ribbon** docked at the left of the screen:



### Close

- Closing the **Ribbon** is not recommended.





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